



KOMPAS-3D V15

User Guide

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Introduction

ASCONE Corporation thanks you for purchasing KOMPAS-3D system and hopes you will find it a reliable and trustworthy partner in your daily work that will significantly expand the scope of tasks addressed with CAD at your enterprise.

The main task solved by the KOMPAS 3D system is modeling products in order to reduce the design period and getting them in production as soon as possible. These goals are achieved due to:

- ▼ fast generation of design and process documentation required for product manufacturing (assembly drawings, BOMs, detailings, etc.)
- ▼ making product geometry available to structural simulation software,
- ▼ making geometry available to development packages of CNC processing stations,
- ▼ creating additional views of products (for example, when creating catalogues, making illustrations for technical documentation, etc.).

The main components of KOMPAS 3D are the three-dimensional solid modeling system, a graphic drafting editor and a BOM (Bill Of Materials) design system.

The three-dimensional solid modeling system is designed for creating three-dimensional parametric models of standalone parts and assemblies that contain both original and standardized structural elements. Parametric technology helps to quickly create models of typical parts based on a once designed prototype. Numerous utilities facilitate solving collateral tasks of design and production line maintenance.

The graphic drafting editor serves to automate design and development activities in various branches. It can be well used in Mechanical Engineering, Architecture, Civil Engineering, making plans and diagrams — wherever you need to develop and produce graphical and text documents.

Graphic and text documents in KOMPASGraphic are created in the same way as in KOMPAS3D, therefore, while working with KOMPASGraphic, one can use the first and second volumes of the present User Guide.

BOM design module can be used together with any of KOMPAS3D components, allowing producing various BOM, schedules and other table documents.

A BOM document can have associative relations with an assembly drawing (its one or multiple sheets) and the three-dimensional assembly model.

KOMPAS 3D functionalities and interface were developed to accommodate typical Engineering Design techniques.

We assure you that you made a right choice by starting collaboration with ASCONE Corporation — one of the leading firms developing Computer Aided Design systems!

How to use the Guide

We hope that acquaintance with description of work in KOMPAS-3D will be useful both for non experienced users and for those who already know the previous system versions (3.x and 4.x, that work in MS DOS environment, as well as 5.x and higher, that work in Windows environment).

The designer who starts to create drawings using the automated design system for the first time, should better first receive basic knowledge on the computer and Windows OS. The contents of the present Guide is intended for a user who already has initial knowledge and skills to work with Windows, namely: work with menus, windows, dialogues, control elements contained in dialogues, etc.

The first volume of the Guide contains information on the system interface, general work methods, as well as detailed description of commands to create and edit objects in graphic documents. The second volume deals with creation of drawings and text documents, describes the document printing procedure, and also service functions. The third volume is devoted to 3D modeling in the system KOMPAS-3D.

An experienced user familiar with Windows and SAPR systems, can ignore the beginning of the Guide, selecting only those chapters that contain description of the concerned function or specific attributes to perform an operation.

Conventions and Abbreviations

To make the text shorter, the following scheme is used to describe the selection of commands from the menu: **Name of a Main Menu Item — Name of the Command Group** (if available) — **Command Name**.

For instance, if the command description writes «...call the command **Highlight — Layer — By Instruction...**», it means that you have to perform this sequence of operations.

1. Select the item **Select** in the Main menu.
2. In the list of commands of the menu **Select** that appears, select the group **Layer**.
3. In the list of layer highlighting methods, that appears, select command **By instruction**.

A similar scheme is used to describe the setup process: **Service — Options... — Name of the Setup Dialogue Tab — Section Name** (group of setup objects) — **Subsection Name** (subgroup of setup objects).

For instance, if the text says: “...call the command **Service — Options — New Documents — Graphic Document — View Arrow...**”, this means that it is necessary to perform such an operational sequence.

1. Select the item **Service** in the Main menu.
2. Select the command **Options...** in the list of commands of the menu **Service** that appears.
3. In the dialogue that appears, activate tab **New Documents**.
4. In the object setup list (located in the left part of the tab) open the **Graphics Document** section.
5. Highlight item **View Arrow**.

If the described command can be called with a button, the icon of this button is to be located in the left field of the paragraph. If the text mentions any buttons, icons, cursors, etc., the corresponding images are also to be located in the left field.

The names of keyboard keys are in angle brackets and italicized. The key combinations are recorded using the "plus" sign, for instance, *<Ctrl> + <F6>*. Such a record means that one should press the key *<Ctrl>*, and then, without releasing it, press the *<F6>* key.

Notes, tips and especially important details are highlighted with horizontal lines and marked with the following signs:



— Note,



— Tip,



— Attention!

Technical support and maintenance

In the case of any installation and operation problems of the KOMPAS system, as well as HASP operation problems, it is recommended to perform the following actions.

1. Refer to the system documentation and try to find information about correcting the encountered errors.
2. Refer to the interactive Help system.
3. If possible, please visit the Internet page of the Technical support service of ASCON company with answers to frequently asked user questions.

Customer technical support site:

<http://ascon.net/support/>

4. If the mentioned sources do not have recommendations regarding the problem you have, get help from the technical support service of your vendor of KOMPAS products (the regional dealer).

Address and phone number of the regional dealer: _____

5. If your vendor could not help you with the problem, please contact ASCON directly.

Saint Petersburg

Phone

(812) 703-39-34

E-mail:

support@ascon.net

Send your correspondence to:

198095, ASCON, P.O. Box 7, Saint Petersburg,

ASCON Internet page:

<http://www.ascon.net>

1. Preliminary Information

1.1. System Installation

1.1.1. System requirements

KOMPAS-3D is intended for use on personal computers based on Intel-compatible platforms running on following operating systems (Table 1.1.1).

Tab. 1.1.1. Supported operating systems

OS	KOMPAS-3D 32-bit version	KOMPAS-3D 64-bit version
MS Windows 8 and later	+	+
MS Windows 7 SP1 and later versions	+	+
MS Windows Vista SP2 and later versions	+	+



The KOMPAS-3D 64-bit version can be installed on a PC with the 64-bit OS version only.

Minimum possible PC configuration for system **installation** and **launch** is the same as for the respective operating systems.

Free disk space required for the installation is specified in Table 1.1.2.

Tab. 1.1.2. Hard disk space required for installation

The installable part of the installation package	KOMPAS-3D 32-bit version	KOMPAS-3D 64-bit version
Pro version	1.5 Gb	1.5 Gb

DVD-ROM drive is required for installation from the distribution disk.

To obtain hard copies of documents use any printers and plotters with available driver for Windows version installed on your PC.



Please note during configuration selection that requirements to computers increase along with tasks complexity growth (drawings saturation, assemblies' complexity).



Operating speed of KOMPAS-3D on a certain computer depends also on PC components' (CPU, RAM etc.) specifications. Ask your vendor about optimum PC configurations.

1.1.2. Installation, launching and deletion of KOMPAS-3D

The KOMPAS-3D system is installed on a computer as follows.

1. Installation of the Basic package. At the same stage, security dongle software is installed.
2. Security dongle connection to computer USB port.



It is recommended to shut down antiviral programs and firewalls (e.g Windows Firewall) during the KOMPAS-3D system installation.

Protection of KOMPAS-3D against unauthorized access is described in Appendix VIII on p. 2121

1.1.2.1. Installation of the Basic Package

In order to install the Basic package onto your computer hard disk please do the following.

1. Start Windows.
2. From the KOMPAS-3D distributive disk, launch the *Setup.exe* file from the following folder: *KOMPAS-3D_V15\KOMPAS-3D*.
3. Follow installation program prompts.

Specify the necessary installation type in **Installation type** window — **Complete** or **Custom** (Fig.1.1.1).

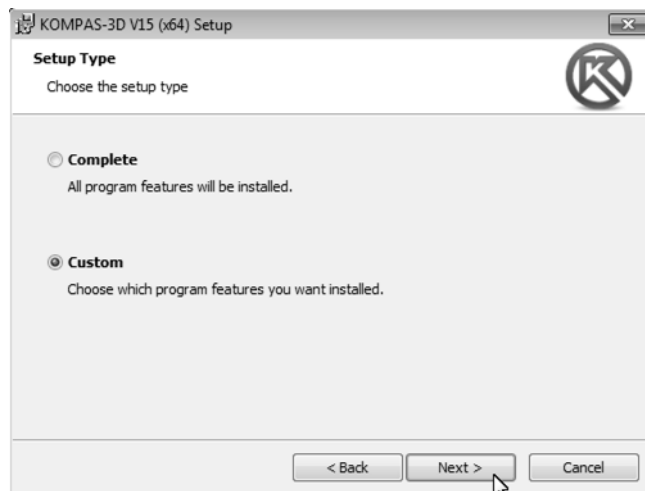


Fig. 1.1.1. Selecting installation type

The first option means installing all system components onto the computer. The second option means selection of components to be installed (fig.1.1.2, 1.1.3).

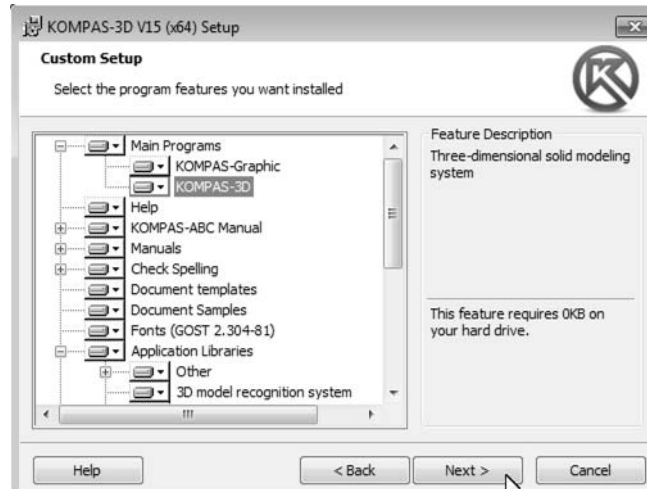


Fig. 1.1.2. KOMPAS-3D component selection

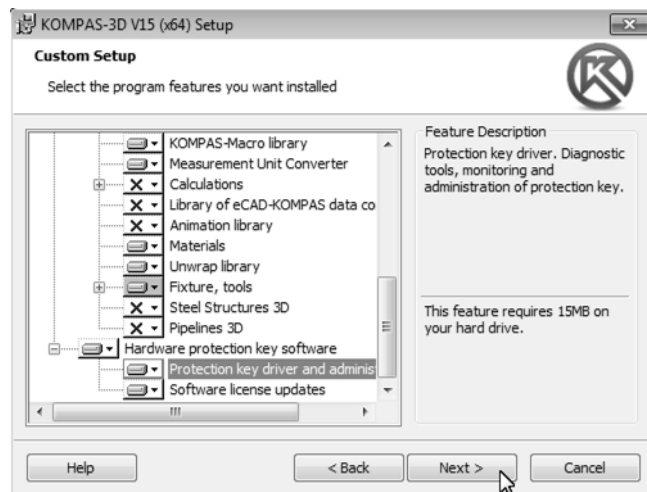


Fig. 1.1.3. Security dongle programs selection

Installation program will copy system files sequentially from the CD to the specified folder on the computer hard disk and then unpack them.

4. If you have to terminate installation before its normal completion for any reason click **Cancel** button.

1.1.2.1.1. Features of KOMPAS-Macro applied library installation

To install KOMPAS-Macro applied library, select this component in KOMPAS system installation window (Fig.1.1.4).

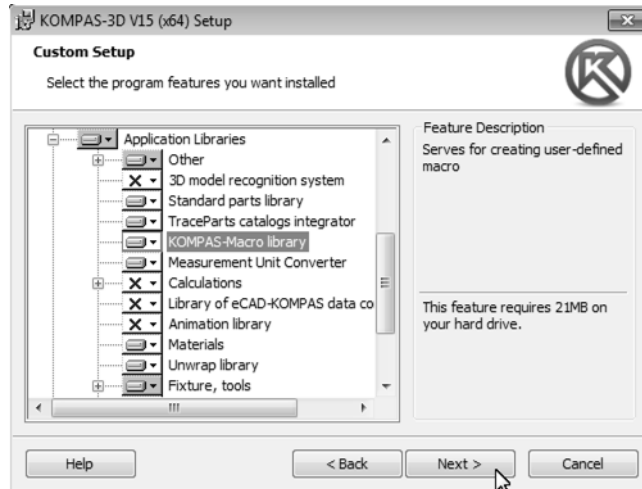


Fig. 1.1.4. System components selection

Python Pywin32 language extension is installed onto computer automatically together with KOMPAS-Macro applied library.

1.1.2.1.2. Preparation to working with the Materials library

In order to connect the “old” database, copy its *MatSort.mdb* file to the *..\\ASCON\\KOMPAS-3D V...\\Libs\\Materials* folder. Then, run the *Materials.exe* file located in the same folder in order to convert the database. Wait until the database is converted and close the library. This conversion is performed only once and will not be repeated afterwards.



For correct operation of the library, users should be granted rights to save in the *..\\ASCON\\KOMPAS-3D V15\\Libs\\Materials* folder.

If the library's database is located in a different folder, the library setup program should be started (*MaterialsCfg.exe*) from the *..\\KOMPAS-3D V15\\Libs\\Materials* folder, and path to the database should be specified.

1.1.2.2. Installation of fonts

Fonts needed for KOMPAS documents execution are installed automatically during KOMPAS-3D installation (see Table 1.1.3):

- ▼ Fonts containing symbols those face corresponds to GOST2.304–81 «ESKD. Drafting typefaces»,
- ▼ fonts containing special symbols.

Font files are copied into the system folder *Fonts*.

Tab. 1.1.3. KOMPAS-3D fonts

	TrueType fonts	Vector fonts
	font name; file name	font name; file name
GOST2.304–81 “ESKD. Fonts typefaces”	GOST type A; <i>gost_a.ttf</i> GOST type B; <i>gost_b.ttf</i> GOST type AU , Unicode font; <i>gost_au.ttf</i> GOST type BU , Unicode font; <i>gost_bu.ttf</i>	GOST type A (plotter); <i>gost_a.fon</i> GOST type B (plotter); <i>gost_b.fon</i>
Special symbols	Symbol type A; <i>symbol_a.ttf</i> Symbol type B; <i>symbol_b.ttf</i>	Symbol type A; <i>symbol_a.fon</i> Symbol type B; <i>symbol_b.fon</i>

1.1.2.3. Possible problem fixing

1.1.2.3.1. System and updates installation, installation recovery

1. During the installation process, we recommend to disable all protective software installed on the machine, such as antiviruses and firewalls (for example, the Windows Firewall).
2. Problems may appear during system installation, update or recovery due to features of users accounts control (UAC). Therefore, if user accounts control is active on the computer, it is recommended to deactivate it for installation, update and recovery period and activate it again after completion of the task.
3. A message about Windows® Installer work termination may appear during KOMPAS-3D installation or configuring. Possible reason is outdated Windows® Installer version (below 4.5). In order to obtain more information and download the Windows® Installer, visit the Microsoft website.

<http://www.microsoft.com/download/en/details.aspx?displaylang=en&id=8483>

Once the Windows® Installer is updated, launch the installation process again.

4. KOMPAS-3D requires Microsoft.NET Framework version 4.0 or a version installed on the machine. Microsoft.NET Framework has to be installed, and the computer should be rebooted prior to starting the KOMPAS-3D installation.

Installation package of Microsoft.NET Framework (*dotNetFx40_Client_x86_x64.exe file*) is available on the KOMPAS-3D distributive disk.

You can also install Microsoft.NET Framework from the Microsoft website:

<http://www.microsoft.com/en-us/download/details.aspx?id=24872>

1.1.2.3.2. Working with applications and libraries Help systems

Operating systems Microsoft Windows Vista and Microsoft Windows 7 contain no program for viewing help in HLP format (files with HLP extension) that was included in previous Windows versions. Therefore, Help systems cannot be used while working with a number of applications and libraries in KOMPAS-3D.

In order to obtain more information and download the HLP help viewer application, visit the Microsoft website.

<http://support.microsoft.com/kb/917607/en-us>

1.1.2.3.3. Displaying KOMPAS documents

In KOMPAS-3D, models are always displayed on the screen using OpenGL. Graphical documents can also be displayed using OpenGL (this is the default method in the Construction configuration). Drawing with the use of OpenGL is possible with hardware acceleration and without it.

If the machine, on which you plan to work with KOMPAS-3D, does not provide enough software and hardware support of OpenGL, display defects and failures may appear when working with KOMPAS-3D.

Possible solutions are presented below.

1. PC upgrade, which may consist of as follows:

- ▼ video card driver update,
- ▼ video card replacement and installation of the respective driver.

Stable operation with built-in video cards (those located on the PC's motherboard) is not guaranteed. We recommend to use video cards with graphical processors.

- ▼ NVIDIA:
 - ▼ Quadro FX 380, 580, 1800 and later;
 - ▼ GeForce of 7XXX series and later;
- ▼ ATI (AMD):
 - ▼ FirePro V5700, V7750, V8700;
 - ▼ FireGL V5600, V7600, V7700, V8600;
 - ▼ Radeon of HD2XXX series and later.

Drivers for video cards can be downloaded from the following websites: NVIDIA (<http://www.nvidia.com/download/index.aspx?lang=en>) and AMD (<http://support.amd.com/en-us/download>). You will have to provide the required information.

2. Disabling hardware acceleration (drawing may slow down).

- ▼ In order to disable hardware acceleration when displaying graphical documents, call the **Service — Parameters — System — Graphical Editor — Image Management** command and disable the **Use Hardware Acceleration** option in the right section of the appearing dialog. Restart KOMPAS-3D.

- ▼ In order to disable hardware acceleration when displaying models, call the **Service — Parameters — System — Model Editor — Image Management** command and disable the **Use Hardware Acceleration** option in the right section of the appearing dialog. Restart KOMPAS-3D.
- 3. Disabling OpenGL (for graphical documents).
In order to disable OpenGL, call the **Service — Parameters — System — Graphical Editor — Image Management** command and disable the **Use OpenGL** option in the right section of the appearing dialog. Restart KOMPAS-3D.

1.1.2.4. Security dongle installation

Security dongle should be inserted in a free USB-port.

No further actions required, since the KOMPAS-3D system checks automatically if a dongle is installed on the computer.

1.1.2.5. Starting KOMPAS-3D

In order to begin working with the system, call **KOMPAS-3D** command from the program group **ASCON — KOMPAS-3DV...** This program group is created automatically during system installation onto hard disk.

Once started, KOMPAS-3D automatically restores its condition at the end of the previous session (loaded documents, window size and location, etc.).



If the previous session has been terminated abnormally (power outage, etc.) with auto save enabled, upon restart the system will restore the opened documents using their temporary copies.

After running KOMPAS-3D for the first time, you will see the main system window without any open document windows. To display remaining interface elements on the screen you shall open a document (see Section 1.3.4.1 on p. 72).



You can also start KOMPAS-3D and simultaneously open a document, for example, by double-clicking the document file's name in the Windows Explorer. In this case, there are several things to take into account.

After starting the system using this method, libraries, that were connected when the previous session was terminated, will not be connected to KOMPAS-3D this time, as well the documents open during the previous session will not be loaded. If, during the session after starting the system in this way, libraries are connected and/or documents are opened, the respective information will not be saved after termination of the current session.

1.1.2.6. Uninstalling KOMPAS-3D

KOMPAS-3D can be uninstalled as normal via the Windows control panel.

All system files are deleted from subfolders of the main system folder during KOMPAS-3D deletion. The resulting empty subfolders are also deleted.

The following files are not deleted (remain in the same folders on the disk):

- ▼ configuration files **.ini*, **.cfg*, **.dsk* и **.prj*,
- ▼ KOMPAS-3D fonts.
- ▼ security dongle driver,
- ▼ files saved in the main folder and its subfolders (document template files, text templates, various libraries etc.).



While working with KOMPAS-3D, users may make changes to any supplied lines style library files (**.lcs*), hatches (**.lhs*), layouts (**.lyt*). Furthermore, users may edit user menu file (*Graphic.pmn*), densities reference file (*Graphic.dns*).

During the KOMPAS-3D deletion, among others, all supplied libraries and files *Graphic.pmn*, *Graphic.dns*, located in *\SYS* subfolder are deleted, no matter if they were edited by the user or not. If these files should be retained for any reason, make their backup copies before KOMPAS-3D deletion.

1.2. System Interface

1.2.1. System window

KOMPAS-3D is a standard Windows application. That is why the workspace you see after launching the system and loading a document is virtually identical to other application windows (Fig. 1.2.1).

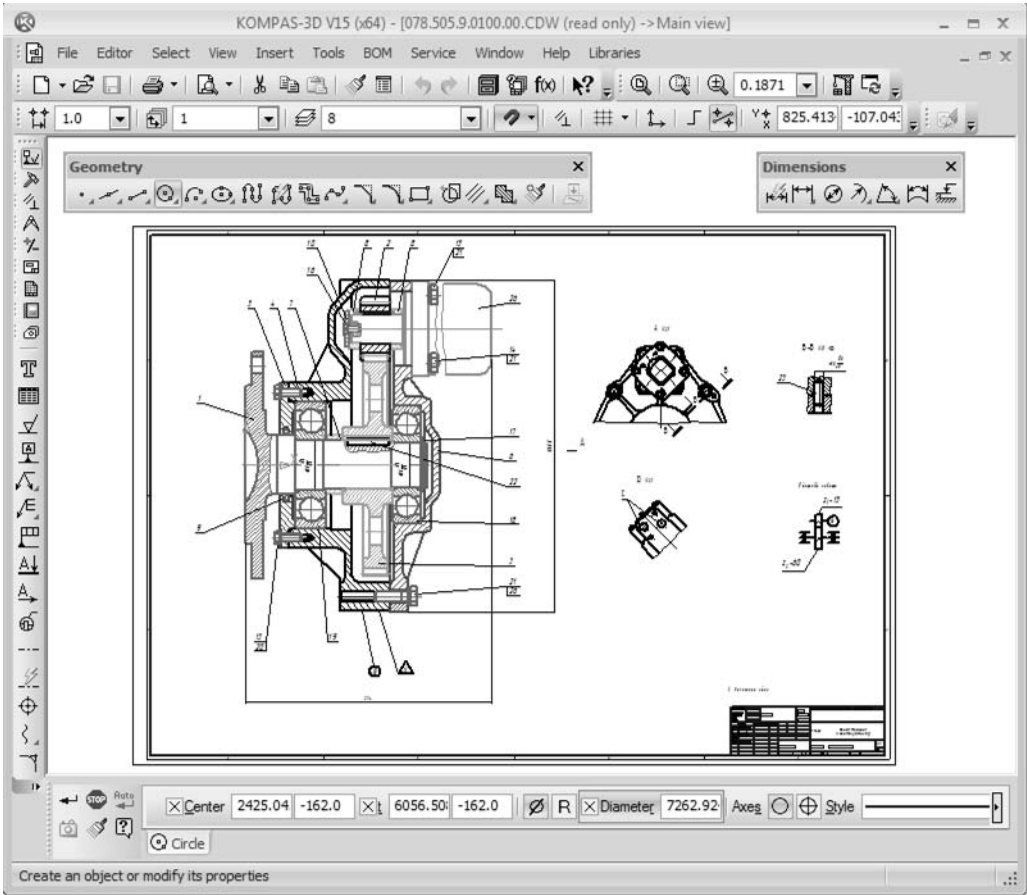


Fig. 1.2.1. KOMPAS-3D interface elements

The functions of the KOMPAS-3D interface elements are represented in Table 1.2.1.

Tab. 1.2.1. KOMPAS-3D interface elements

Name	Description
Header	Contains name, system version number, current document* name, system menu button and the system window management buttons.
Main menu	Serves to call system commands. Contains names of menu pages. The Main Menu contents depend on the current document type and system work mode.
Toolbars	Contain system command calling buttons.
Compact Toolbar	Contains several toolbars and the buttons to switch between them (for more information, see Section 1.2.2.1.1 on p. 53). Compact bar contents depend on the active document type.
Variables window	Serves to work with variables and expressions (for more information, see Section 7.1 on p. 1569).
Library Manager	Serves to work with KOMPAS-libraries. For details on Library Manager, see Section 12.1.1 on p. 2035.
Properties bar	Use this bar to set up an object when during its creation or editing (more details can be found in Section 1.2.2.1 on p. 52).
Line of messages	Contains system messages that are relevant to the current command or to a working window element pointed at by the cursor.
Tree model	Displays the model (drawing) creation order and links between its elements and components. Can only be placed inside the Document window (for more — see Sections 2.1.2 on p. 104 and 3.6.1.1 on p. 1207) .

* In order to choose a variant of file name display (full or short), call the **Service — Parameters...** command. — **System — General — File names displaying** command.

The Header and the System main menu are permanently displayed on the screen. Display of remaining interface elements is controlled by a user. Commands for enabling or disabling these elements can be found in the **View — Toolbars** menu.

1.2.2. Bars (panels) and windows

1.2.2.1. Toolbars

All toolbars presented in the KOMPAS-3D window by default, can be subdivided into two groups.

- ▼ The first group contains **Standard**, **View** and **Current Status** bars. These bars contain the buttons to call the commands to work with a document as a whole (saving, scale editing commands, etc.). However, only some commands are universal and can be used when working with a document of any type. Therefore, depending on the type of the current document the contents of the first group bars are changing: buttons for command that are not required at the moment are deleted while buttons for required command are added.
- ▼ The second group contain all remaining toolbars. They contain buttons for calling commands for creating and editing objects related to a certain document type. Buttons on bars are grouped by function and they form an advanced command bars (see Section 1.2.2.1.2).
The second group bars are grouped into the system compact bars for every document type (for more about compact bars see Section 1.2.2.1.1).
Object types are determined by a document type (for example, a drawing can not contain bodies, a model can not contain views). Therefore, when switching to another type of document, components of the compact bar, i.e. toolbars contained in it, will change completely.
Display of bars is enabling and disabling by commands that are located in the **View – Toolbars** command submenu.



After connecting libraries to KOMPAS-3D, their toolbars become available.

A bar displaying of which is enabled can be in the “floating” or fixed state. “Floating” bar can be located in any place of screen, fixed— only inside KOMPAS-3D window with docking at some of its borders.

To fix a bar next to a border of window, drag with the mouse at its title to this border. To return a bar to a “floating” state, perform a reverse action, namely, drag it in the window center direction.



If the bar is fixed, then instead of the title there is an embossed line by the left or top end of the bar.



For quick switching between “floating” and fixed bar state, you can double click its title.

If necessary, you can create user-defined toolbars and place on them buttons of any commands (see Section 9.4.2.2 on p. 1874).

1.2.2.1.1. Compact bars

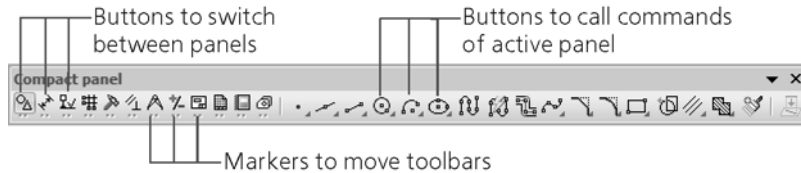


Fig. 1.2.2. System compact bar used when working with a graphic document

The compact bar contains a number of toolbars represented by buttons for switching between them and by buttons for calling commands of the active bar. Activation of a certain toolbar is performed using switch buttons.

By default KOMPAS-3D, the system compact bar is displayed in window. It contains toolbars for creating and editing objects related to the document of a given type.

You can change contents of the system compact bar. Next to the switch buttons there are drag markers. To remove a toolbar from the system compact bar, “drag” its respective marker by the mouse outside the system compact bar.

Release the mouse button. The selected toolbar will appear on the screen. Its respective button will disappear on the system compact bar.

Any toolbars, except **Standard**, **View**, **Current Status** and also compact bars, can be grouped into a user-defined compact bar.

To do this, press and hold the <Alt> key and then “drag” one bar to another with the mouse at its title. When, upon overlapping the panes, the “+” sign appears next to the cursor, release the mouse button, and then release the <Alt> key. The user-defined compact bar will be created. It is automatically assigned with name “Compact bar № N”, where N is a sequential number of user-defined compact bar.

To restore or add a toolbar in the compact bar, “drag” the toolbar’s heading so that it “overlaps” the compact bar, keeping the <Alt> button pressed. After the “+” sign appearance, release mouse button and key. Toolbar will be included into the compact bar.

1.2.2.1.2. Advanced Command Bars

Buttons for calling commands are grouped by their purpose and represented on the toolbar by the button of a command of the group. If a command’s button is clicked and kept clicked, an **expanded bar** appears next to the button comprising all commands of a given group. For example, an advanced bar called by the **Arcs** button of the **Geometry** bar contain different methods of arcs creation: by three points, tangent to curve and other.

The buttons that produce an additional command bar are marked by a small black triangle in the lower right corner (Fig. 1.2.3, a).

The expanded command bar can be transformed into a separate bar with a generalized heading, for example **Arcs** (Fig. , b).1.2.3

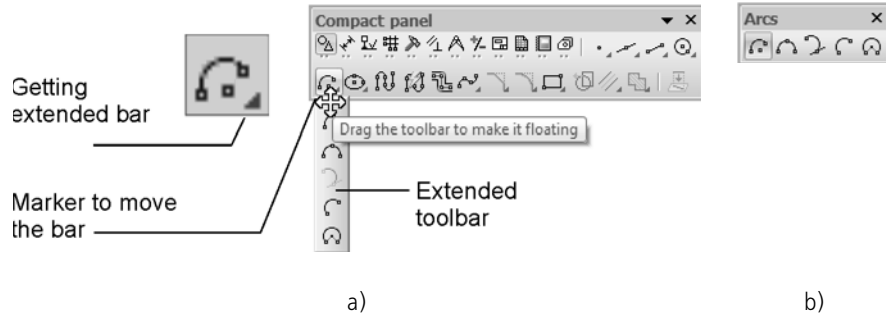



Fig. 1.2.3. Extended Command Panel:
a) within the toolbar, b) separate

 To detach the extended panel from the toolbar, perform the following actions. On the command button: Call the extended panel, keep the left mouse button pressed, move the cursor to the drag marker— “wavy” line at the panel edge. After the cursor turns into a four-sided arrow, release the mouse button— the extended panel shall stay on the screen. Click the mouse left button and drag the panel to any place on you screen.

To delete a separate expanded bar from the screen, close it. Create it again, if you want it back. Individual expanded bars cannot be included into compact bars; you cannot change the button set and their order in expanded bars.



Commands on separated expanded bars and on toolbars are equally available for calling. At the same time, expanded bars can be used without having a toolbar on the screen which was used to create them.

1.2.2.2. Windows

A window is widely used in KOMPAS-3D as a control element. This is a specific section of the screen. You can modify the size and position of this section. The main part of the window is a field for information entry and output. Windows can have their own toolbars. Examples of windows: Variables Window, Library Manager, **Properties** window, **Numbering** window, **Technical requirements** window, **document tree** (for assembly) window, and an additional Tree window. Some windows, for example, the **Document Tree** window, are parts of documents. Therefore, they can only be positioned inside the document window. Other windows (like the **Properties** window) belong to the whole KOMPAS-3D system and can be positioned in any part of the screen.



Property bar is also a window (see Section 1.4.2.1 on p. 82).

1.2.2.3. Controlling the state of bars and windows

Bars and windows may be displayed on the screen in one of the following two states: “floating” or fixed.



If necessary, you can fix positions of both floating and fixed bars and windows (see Section 1.2.2.3.5).

Bars can be fixed beside a border of a system window. To fix a bar, drag with the mouse at its title to the required border. To return a bar to a “floating” state, perform a reverse action, namely, drag it in the system window center direction.

Windows that belong to a document (such as **Document Tree** and **Technical Specifications**) can be fixed next to the vertical border of the document’s window. Such windows can be fixed in the same way as bars, i.e. by “dragging” their header with the mouse.

Other windows belong to the whole KOMPAS-3D system and can be fixed next to any border of a KOMPAS-3D window (**Numbering** and **Properties** windows can be fixed only next to vertical borders). The windows can be fixed in the following way:

1. Hover the mouse's cursor over the window's header, press the left button and move the mouse.

In the center of the KOMPAS-3D window and beside those its borders where the window being moved can be fixed, control elements for selection of its position appear.

2. Move the mouse button to the required control element.

In the KOMPAS-3D window, the border area, which will be occupied by the window being moved, will be highlighted by color.

3. If the position of the window is what you need, release the mouse button.

The window is fixed next to the chosen border.

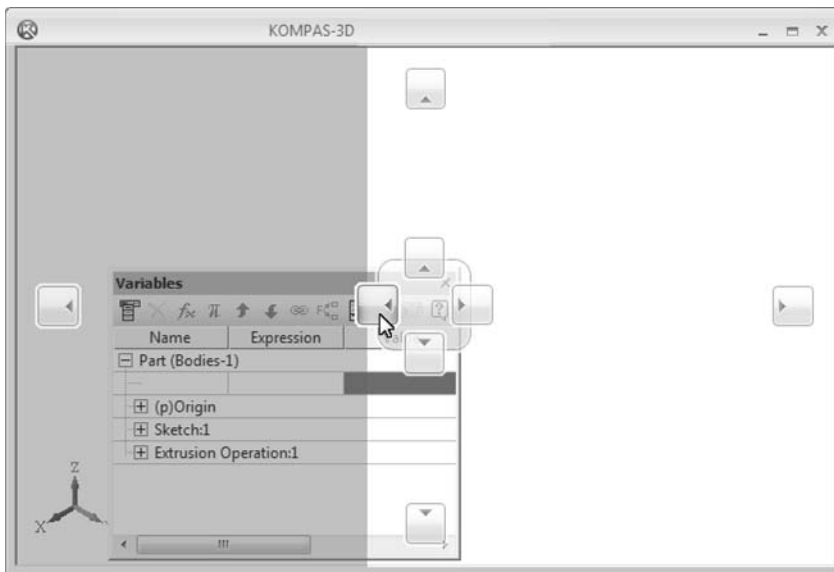


Fig. 1.2.4. Fixing the Variables Window next to the left border of the system window



To lock the property bar near the required system window boundary, you can also use **Location – Up/Down/Left/Right** commands from the contextual menu of the Bar.



For quick switching between “floating” and fixed bar or window state, you can double-click its header.

1.2.2.3.1. Combining windows

Windows with similar fixation variants (to any KOMPAS-3D window border or to vertical borders only) can be combined. The windows can be combined in the following way:

1. Hover the mouse cursor over the header of one window, press the left mouse button and move the mouse, so that the cursor is positioned within the borders of the second window.

Control elements for selection of the first window's position will appear in the center of the second window (Fig. 1.2.5, a).

2. Move the mouse button to the required control element. In the second window, the area to be occupied by the first window (the one being moved) will be highlighted in gray color.

- ▼ The central control element makes it possible to combine windows by overlapping them; in this case, tabs will be formed (Fig. 1.2.5,b).

- ▼ Other controls make it possible to combine windows by dividing the area of the second window vertically or horizontally (Fig. 1.2.5, c).

3. If the position of the window is what you need, release the mouse button.

The first and the second windows are combined forming a single window.

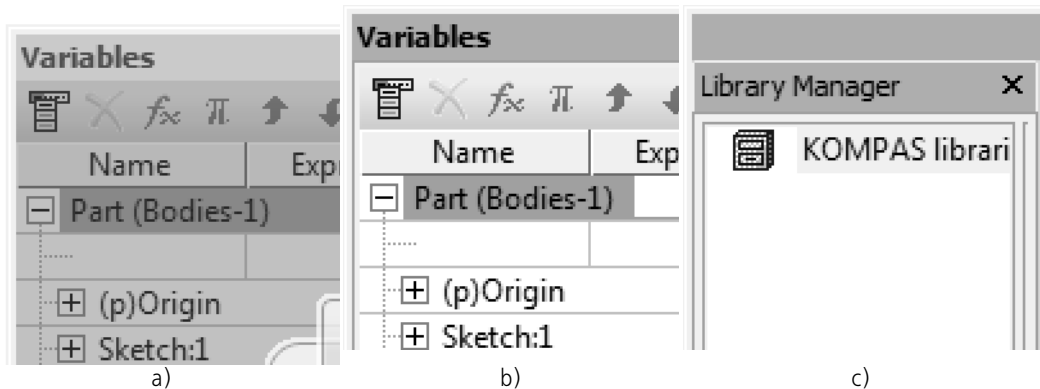


Fig. 1.2.5. Combining windows:

a) combination variant selection, b) window with tabs, c) window divided vertically



In order to overlap windows with forming tabs, you can also “drag” the first window to the second window's header.

1.2.2.3.2. Combining the Properties window with the Property bar

The purposes of the **Properties** window and the Property bar makes it impossible to use them at the same time: the **Properties** window is used for changing properties of selected objects, while the Property bar is designed for specifying properties of an object being created (edited). To save screen space, you can combine the **Properties** window and the Property bar in a single window. You can do this in one of the following ways:

- ▼ by calling the **Location — Combine with the Properties window** command from the Property bar's context menu,
- ▼ or by calling the **Combine with the Property bar** command from the context menu of the **Properties** window's header.

The combined window is positioned vertically; it can be fixed beside the vertical border of a KOMPAS-3D window.

Depending on the currently performed operation, the combined window displays:

- ▼ the Property bar — during creation (editing) of an object,
- ▼ the **Properties** window displaying objects' properties — if selected objects are present in the window,
- ▼ the empty **Properties** window — if there are no selected objects.

If you need to cancel the combining:

- ▼ if the combined window displays the Property Bar, call the **Location — Combine with the Properties window** command from the context menu or disable the display of the Property Bar and enable it again;
- ▼ if the combined window displays **Properties** window, call the **Combine with the Property bar** command from the context menu in its header or disable the display of the **Properties** window and enable it again.

1.2.2.3.3. Automatic minimization of windows

A window (including windows formed by overlapping or combining with other windows), fixed beside a system window or document border, can be automatically minimized to the border. This allows to use working field more effectively: if you do not work with a window, it will disappear automatically, and its label will be visible on the border of the system window or document with its name displayed. In order to display the minimized window on the screen and continue working with it, place the cursor in the label's area.



In order to enable/disable automatic minimization of the window, use the **Fix** button in its header.


1.2.2.3.4. Commands for controlling a window's status

To manage a window's status, use contextual menu commands in its header (Tab. 1.2.2).



If the Property bar is docked horizontally (by the top or bottom border of the system window), then instead of the title there is an embossed vertical line by the left end of the Bar (see Fig. 1.2.1 on p. 51).

Tab. 1.2.2. Window status management commands

Name	Description
Floating	Switches a docked window into the “floating” state. The Floating command is not available, if automatic minimization of a window is enabled, or if floating windows' position is fixed.
Fixed	Docks a “floating” window. Once the command is called, the window will be fixed near the border where it was positioned before becoming the “floating” one. The Fixed command is not available, if automatic minimization of a window is enabled, or if docked windows' position is fixed.
Minimize	Enables automatic hiding of a “fixed” window. To disable the auto hide, call the command again. Hiding a docked window can be controlled also by the Fixation button located in the window's header. The button image will change after the clicking, which indicates the auto hide mode:  <ul style="list-style-type: none"> — window's automatic minimization is enabled, — window's automatic minimization is disabled.
Close	Closes a window. After closing the window, it will disappear from the screen. To display it again, you will need to call the window display command.

If the **Technical Specifications** or **Document Tree** window is in the “floating” state, it can be displayed with different transparency degrees. In order to adjust transparency, call the **Transparency — N%** command from the window's header (Fig. 1.2.6). If you left-click a transparent window, it will become non-transparent.

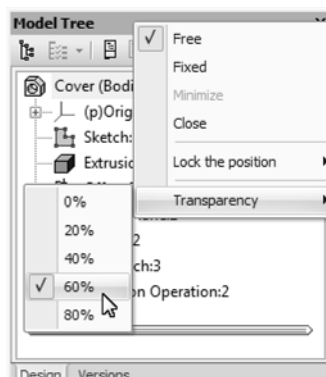


Fig. 1.2.6. Example of window transparency adjustment



On a PC with Microsoft Windows XP, there is no **Transparency** command in the context menu of the **Technical Specifications** and **Document Tree** “floating” windows' headers. The **Technical Specifications** and **Document Tree** windows are displayed as non-transparent.

1.2.2.3.5. Fixation of bars and windows

You can fix positions of bars and windows. For this purpose, call the **View — Toolbars — Fix position** command. The submenu with the following commands will be displayed: **Floating bars**, **Fixed bars**, **Floating windows** and **Fixed windows**. Commands can also be called from the window's header context menu or from the menu of the arrow button on the bar's header (Fig. 1.2.7).



Fig. 1.2.7. Example of calling the command

As a result of fixing the position:

- ▼ the following operations will be blocked for bars and windows:
 - ▼ movement
 - ▼ resizing
 - ▼ switching between the “floating” and fixed states by double-clicking the header;
- ▼ the following operations will be blocked bars:
 - ▼ modification of commands' order and set,
 - ▼ separation of expanded command bars and menus (for fixed bars);
- ▼ the following context menu commands will become unavailable for windows:
 - ▼ **Floating**,
 - ▼ **Fixed**.

If a window is formed by combining other windows, fixing its position will block their borders' movement (in case of external borders, only if the window is fixed).



During interface setup (see Section 9.4.2 on p. 1872), bars' fixation will be temporarily disabled. You can modify the order of commands, the set of command, as well as control their display mode.



If you try to move a fixed bar or window, the cursor will be transformed into a warning symbol.

In order to temporarily disable fixation of bars and windows, press and hold the **<Ctrl>** key.

1.2.3. Document Window Management

KOMPAS-3D allows working simultaneously with several different documents.

Any document, whether a newly created one or opened for editing, is displayed in a separate window.

If necessary, you can work with the same graphic document or model document in several windows. In order to open an extra document window, call the **New Document Window** command from the **Window** menu or from the context menu of the document's tab. Different parts of a document can be displayed in different windows to different scales.

Each window has a title that contains the name of the displayed document, the system menu button and the window management buttons.

The techniques of using windows are standard (dragging a window by the header, resizing the window, etc.) and identical to those of other Windows applications. To conveniently arrange document windows on the screen, use the commands **Cascade** and **Tile...** in the **Window** menu.

Any document window can be minimized to a button, or icon. This can be convenient in the case, when the document is needed only temporarily, and the user would prefer not to close it at all. To orderly place document icons (which are usually displayed in the bottom part of the main KOMPAS-3D window), use the **Window — Arrange Icons** command.

To cyclically switch through document windows, you can use the <Ctrl>+<F6> keys combination, and for random access — the **Window** menu commands. Additionally, you can use document tabs to select the current document (see Section 1.2.3.1 on p. 61).

Managing documents and windows is also possible in the **Windows** dialog called by the command **Window — All Windows**.

The view window of the dialog lists all windows open during the current session.

Select one or several documents in this list and perform one of the following actions using the relevant buttons:

- ▼ activate (possible when one document is selected),
- ▼ save,
- ▼ close,
- ▼ cascade,
- ▼ tile horizontally,
- ▼ tile vertically,
- ▼ minimize.



The **Cascade**, **Horizontally**, **Vertically**, and **Minimize** buttons are available in the **Windows** dialog, if displaying of tabs is disabled (see Section 1.2.3.1 on p. 61).

1.2.3.1. Document tabs

The main purpose of the tabs is to enable quick switching between document windows. The document's file names are displayed in the document's tabs (Fig. 1.2.8).

You can enable or disable tabs using the **Window — Display Tabs** command. If a check mark is displayed next to this command in the menu, the tabs are displayed.

Location and appearance of tabs, document file names format (full or short) are adjusted in the document tab setup dialog (see Section 9.1.3.7 on p. 1684). You can also enable and disable tabs in the dialog.



Tabs can be used only with maximized document windows. Therefore, when enabling tabs display, all windows maximize automatically. The **Minimize** and **Restore** system buttons for windows will disappear, while the **Close** button will be placed on the right on the current document's tab or in the tab row depending on the setup (see Section 9.1.3.7 on p. 1684).

In order to enable a document's window, click its tab and select the required document in the list which is displayed after clicking the arrow button in the right part of the tab row (Fig. 1.2.8).



Fig. 1.2.8. Document Tabs

The document tab's context menu contains the following commands:

- ▼ **Save;**
- ▼ **Save as;**
- ▼ **Close window;**
- ▼ **Close all other windows;**
- ▼ **Close all;**
- ▼ **Close other document windows;**
- ▼ **New document windows;**
- ▼ **Open folder with file;**
- ▼ **Copy full name of file;**
- ▼ **Customize...;**
- ▼ **All windows...**



In order to close a window, double-click the left mouse button or click the middle mouse button (wheel) once on the document's tab.



In order to create a new document, double-click the left mouse button on a free space of the tab row.

If displaying document tabs is enabled and more than one window is opened, the workspace of the main system window can be divided into several parts (see Section 1.2.3.2).

1.2.3.2. Workspace

You can divide the workspace of the main system window into several parts. One or several document windows can be in each of them.

Splitting is possible when more than one window is open (as well as for one document) and tabs are shown.

To split the workspace vertically, “drag” any tab by the mouse into a document window or to the right-hand border of the workspace. The workspace will be divided in two parts. The document window, whose tab was moved, will be on the right-hand part, while the other windows — on the left part.

The left part can be divided in two more parts in the same way, provided that it still has more than one window.

Horizontal splitting of the workspace can be done in the same way. For this purpose, “drag” any tab to the lower border of the workspace.

You can move windows across sections by “dragging” the tabs by the mouse from one section to another.

To change sections width, “drag” their borders by the mouse.

To return to the original view of the work area (i.e., put all of its parts back together), “drag” to any part of the work area the tabs of all windows that are outside this part.

1.2.4. Context menus and context bars

The context menu appears when you right-click the mouse. The menu contents depend on the object on which the cursor rested when you right-clicked and on the action being performed. The menu contains the most typical commands at the given project stage (Fig. 1.2.9).

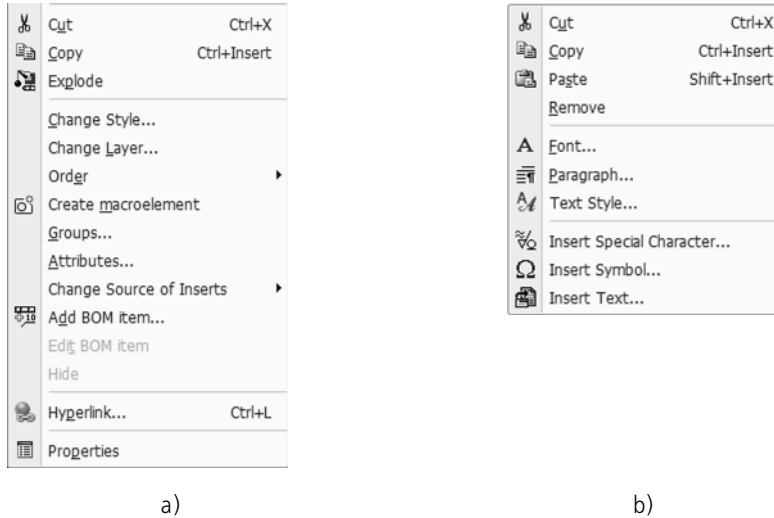


Fig. 1.2.9. Contextual menu:
 a) for several selected geometric objects,
 b) for the selected text fragment

The context toolbar is displayed:

- ▼ when objects are selected in the document window,
- ▼ when objects are selected in the Drawing Tree or in the Model Tree,
- ▼ At a mouse click in any free space of the document
- ▼ when the context menu is called, except when calling the context menu during some command execution.

The context bar includes the most frequently used command buttons. The bar content depends on the type of the selected object, the document type and the current work mode.



The context bar will not appear when selecting the following objects:

- ▼ designations for civil engineering
- ▼ Designations for mechanical engineering, except for leaders and position designation,
- ▼ Bent radial dimension and level dimension
- ▼ OLE-insert.

In Fig. 1.2.10, you can see some context bars.



Fig. 1.2.10. The context toolbar when you select: a) a geometric object in a drawing or fragment, b) a body face in an assembly, c) a text fragment in a text document, d) a geometric object in a 3D element sketch

You can configure contents of the context bar. This can be done with the **Customizing Interface** button in the right part of the toolbar (Fig. 1.2.11).

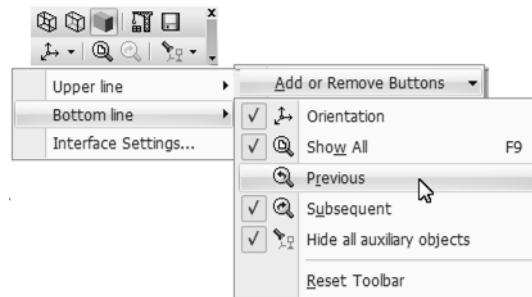


Fig. 1.2.11. Configuring contents of the context bar

Besides, you can set up display parameters for the context toolbar (in particular, it can be disabled in documents of one or another type). To do this, call the **Service — Parameters... — System — General — Context bar**.

The context bar appearing at context menu call does not include the **Customizing interface** buttons.

The context bar looks semi-transparent, but becomes non-transparent when pointing the cursor on it. The context bar disappears when moving the cursor out of the selected object or calling any command.

The convenience of context menu and toolbars is provided for by grouping together commands scattered across various sections of the main menu but frequently used in work. Moreover, commands are activated quicker because the context menu is displayed in the screen area, where it was called by a mouse click, and the context toolbar — near the cursor.

Press the <Esc> key to close the context menu and/or context bar.

1.2.5. Mode icons in the document window

Mode icons serve to inform the user of the current mode of the object (model, graphic object, BOM item, etc.) and to exit from the mode.

An icon that corresponds to the mode button on the panel and which appears in the document window when the system switches to a particular mode and disappears when leaving it. Several icons can simultaneously display in the Document window.

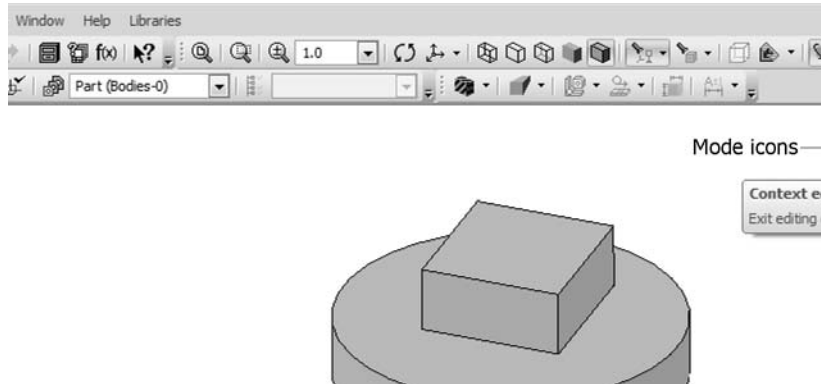


Fig. 1.2.12. Mode icon displaying

The **Show mode icons** command from the **View** window serves to control the appearance of icons on the screen, while commands from the icon's context menu are used for location in the document window.

Icons are displayed for the following operating modes:

- ▼ editing graphical macroelement,
- ▼ editing technical requirements,
- ▼ editing BOM items,
- ▼ sketch,
- ▼ flat pattern,
- ▼ recalculating of dimensions based on tolerances,
- ▼ context editing,
- ▼ components exploding,
- ▼ model section,
- ▼ preview mode before printing,
- ▼ editing the title block table.

In order to exit the mode, you will need to click the icon.

1.3. KOMPAS-3D Documents

1.3.1. Document Types

The type of the document created in the KOMPAS-3D system, depends on the type of information stored in that document.

Each document type has a file name extension and an individual icon.

1.3.1.1. 3D Models

Part

Part — model of the product manufactured using a homogeneous material without assembly operations.

A part file has the *m3d* extension.

Assembly

An **Assembly** is a model of a product consisting of several parts with a certain mutual position.

The assembly can include other assemblies (subassemblies) and standard products.

An assembly file has the *a3d* extension.

Technological assembly

The **Technological assembly** is an assembly that contains technological data, for example, the result of model size recalculation taking account of tolerances, technological objects (center holes, mounting holes, and so on), technological models (lunettes, centers, instruments, and other equipment).

The technological assembly's file has the following extension: *t3d*.

1.3.1.2. Graphic documents

Drawing

A main type of graphic document in KOMPAS-3D — **drawing**. A drawing contains a graphical image of the item, a title block, a border, sometimes — advanced layout elements (a symbol of unspecified surface finish, the specifications, etc.). A KOMPAS-3D drawing may contain one or several sheets. For each page, you can set format, multiplicity, orientation and other properties. Aside from drawings (in terms of USDD — the Unified System of Design Documentation), the file of the drawing KOMPAS-3D may contain diagrams, posters, and other graphical documents.

A drawing file has the *cdw* extension.

Fragment

A special type of graphic document in KOMPAS-3D — **fragment**. Fragment differs from a drawing because of absence of border, title block and other document layout elements. It is used for storing drafts that do not need to be laid out as a separate sheet (such as sketches, roughs, etc.). Additionally, you can use fragments to store standard solutions for further use in other documents.

A fragment file has the *frw* extension.

1.3.1.3. Text documents, BOMs

Bill of Materials (BOM)

Specification is a document with information about the contents of assembly represented in the form of a table. BOM uses a format frame and a title block. It is often multipage.

A BOM file has the *spw* extension.

Text Document

The document that contains mostly text info — **text document**. Text document uses a format frame and a title block. It is often multipage. You can create explanatory notes, notifications, technical specifications and so on, in a text document.

A text document file has the *kdw* extension.

1.3.2. Coordinate systems and measurement units in documents

1.3.2.1. Coordinate Systems

When working with KOMPAS-3D the standard right-hand oriented Cartesian coordinate systems are used.

In every graphic and model document there is an absolute coordinate system. It is not possible to delete it from the document.

In a 3D model, the coordinate system defines coordinate planes. The coordinate system is conventionally displayed on the screen as three orthogonal segments, while planes are represented as rectangles lying in these planes. If needed, local coordinate systems may be created. For detailed description of coordinate systems in 3D models, see Section 2.8.3 on p. 595.

The coordinate system in a graphical document is located in a plane parallel to the screen, and is displayed as two orthogonal arrows. If needed, local coordinate systems may be created. For detailed description of coordinate systems in graphics documents, see Section 3.1.5 on p. 887.

1.3.2.2. Measurement Units

Standard metric system of measurements is used in KOMPAS-3D.

Default length unit — millimeter.

These units are used to set and display linear parameters (for example, length of the segment, circle radius, offset value) on the property bar, values in dimension texts of the dimensions, cursor coordinates, and so on.

You can set other length unit for graphic documents— centimeter or meter (see Section 9.2.6.2 on p. 1760).

Default angular unit— degree.

Angular parameters are set and displayed in these units (for example, arc opening angle or rotation angle of an object copy) on the Properties bar. You can set other angular units for the Properties bar fields— degrees, minutes and seconds, radians (see Section 9.1.2.2 on p. 1668).

Values in dimension texts of angle dimensions are expressed in degrees by default. You can set another unit— degrees and minutes or degrees, minutes and seconds (see Section 9.2.6.10.1 on p. 1771).

When calculating mass-inertia properties, you can control the representation of results choosing the required units directly in the measurement process (kilograms or grams for mass; millimeters, centimeters, decimeters, or meters for length).



When drawing in KOMPAS-3D, **the user always uses real dimensions of objects** (1:1 scale), while the image's positioning on the drawing of the required format is ensured by choosing the appropriate scale view (see Section 3.5.4.1 on p. 1161).

1.3.3. Creating and saving documents

1.3.3.1. Creating Document



To create a new document, click **File — Create**.

A document creation dialog will appear (Fig. 1.3.1).



Fig. 1.3.1. Dialog to Create a Macroelement

On the **Templates** tab you can select the desired template for the new document.

If you do not need a template, then select the document type on the **New Documents** tab.

Click the **OK** button to create a document of a specific type or by a specific template.

Another way to create a new document is selecting it from the **New** menu button. (Fig. 1.3.2).

The commands of this menu can be placed as buttons on a separate bar and then moved to any convenient location. To do this, drag the **New** button with the mouse at its title in any direction. The **New Document** toolbar will be created (Fig. 1.3.3).

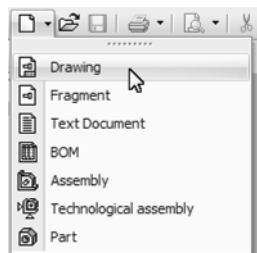


Fig. 1.3.2.

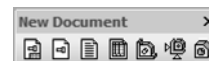


Fig. 1.3.3.



Note the difference between the **New Document** bar and the other toolbars: you cannot change the composition or the order of buttons on them.

Default templates and formats are used to create new documents. If necessary, you can customize the template and the format of an active document (see Sections 3.5.2.1 on p. 1151 and 4.1.5.2 on p. 1343).

1.3.3.2. Saving Document



To save a document to a file on your hard drive, call the **File— Save** command.

The document will be saved automatically to the same folder and file as the previous time. In other words, the file created last time you saved a document will be replaced with a new file containing the document with changes.

If necessary, you can save the document under a different name or in a different folder (see Section 1.3.3.2.2).

1.3.3.2.1. Saving a Document for the First Time



To save the document, click **File — Save**.

The Save file wizard dialog will appear. Specify the folder to which you would like to save the document, enter the filename, and click **Save**.

By default, the system suggests a file extension matching the document type.



Do not change the standard document file extension unless it is absolutely necessary, as this will make it difficult to find the file in the future (it will not be displayed in the list of documents of this type).

When you try to save a document to the existing file, the request will pop up to confirm rewriting (replacement of the old document with a new one). If the existing document is open at the moment, it cannot be replaced with a new one; the write failure message will pop up.

After the File Save dialog closes, the Document Summary dialog will appear (see Section 1.3.5 on p. 73).

1.3.3.2.2. Saving Document under a Different Name

Sometimes you may need to save, a document after editing, while keeping the old file version unchanged. In this case, save the document under a different name or in another location on the disk.

Call the command **File—Save As...**

The further steps are identical to those when saving a document for the first time (see Section 1.3.3.2.1). The only difference is that the Document Summary dialog will not appear.

In addition, the **Save as...** command allows you to save KOMPAS-documents as templates for new documents, convert KOMPAS-documents to other formats, and, when working with 3D models, you can save a part in an assembly file and an assembly - in a part file. To save the document as a template, for conversion or to change model type select the required line from the **File Type** list.

1.3.3.2.3. Saving All Documents

To save all open documents at once, click **File — Save All**.

When saving documents that are being written on a disk for the first time, a standard dialog will appear, in which you need to define the file name and the storage location. After this dialog closes, the Document Summary dialog will appear.

1.3.4. Opening and closing documents

1.3.4.1. Opening Document



To open an existing document, click **File — Open**.

In the dialog that appears, select the document type, specify the filename, and click **Open**.

You can also open a KOMPAS-3D document using Windows Explorer. To do so, in Windows Explorer double-click the file you need to open or right-click the file and select **Open** in the context menu.

If you attempt to open a document, which has already been loaded for processing, KOMPAS-3D will not open it again, but rather activate this document window.

If you have recently edited a document and then closed it, you can reopen it faster. In the bottom part of the **File** menu, a list of the last nine documents, with which you had worked, is displayed. Indeed, this is a list of documents in the order in which those were closed. To open the desired document, simply select its name in the menu.



You can configure the number of recently opened files that are displayed in the **File** menu. To do this, call the **Service — Parameters... — System — General — File names displaying** command.

You can open documents from one folder on by one without calling the **File— Open** command or using Windows Explorer.



To switch between documents, **Previous document** and **Following document** commands are used.



The preceding (following) document with respect to a given document is the document contained in the same folder as the given document and preceding (following) it, if sorted by file type.

When calling the **Previous document** command, the preceding document with respect to the current one is opened in the active window. If the **Following document** command is called, the next document is opened.

Thus, if the current document had been opened from a window, this document will be closed when **Previous document** and **Following document** commands are called.

If the document opened in active window is the last in the file list sorted by type, **Next Document** command is unavailable. If this document is the first one, the **Previous document** command is unavailable.



The **Previous Document** and the **Next Document** are the viewer application commands. In order to use these commands in KOMPAS-3D, add them to any menu or bar (see Section 9.4.2.1 on p. 1872).

1.3.4.2. Closing a document

To close a document, call the **File — Close** command.

If any changes to the document have not been saved, a prompt will be displayed about saving the document being closed.

If the document was displayed in several different windows, all those windows will close.

To close only one document window and leave the remaining windows open, double-click the system menu button of the window you want to close.

1.3.5. Document Information

The document information is various reference data about this document, which are stored with it, such as the author's name, creation date and time, attributes, etc. You can view and edit these data in the **Document Information** dialog (Fig. 1.3.4).

Click **File — Document Information...** to open this dialog. The dialog appears automatically when you save a document for the first time. The document summary dialog control elements are presented in Table 1.3.1.

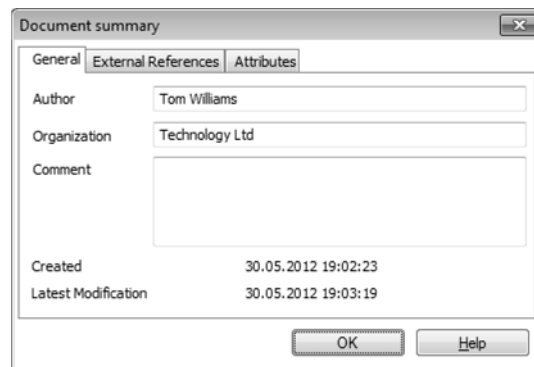


Fig. 1.3.4. Document Info Dialog

Tab. 1.3.1. Document Info Dialog

Element	Description
Tab General Information	Contains general document information.
Author	Author's name.
Organization	Company name.
Comment	Document comments. A comment can span multiple lines.
Created	Document creation date and time*.
Last Modification	Date and time of last document saving*.

Tab. 1.3.1. Document Info Dialog

Element	Description
Tab External Link	<p>The viewing pane of this tab lists all external files referenced by this document, and with which it is linked:</p> <ul style="list-style-type: none"> ▼ Format libraries ▼ Fragment libraries ▼ Style libraries ▼ Attribute type libraries, ▼ BOM documents attached to the current assembly or drawing, ▼ Assemblies or assembly drawing sheets attached to the current BOM, ▼ Documents attached to the BOM objects in the current document, ▼ Documents, whose BOMs the current document is connected to, ▼ Inserted fragments ▼ Files of parts related with associative views ▼ files of the parts and assemblies inserted in the model, ▼ E-mail addresses and external hyperlinked files.
Whole Path	<p>If the full path and filename do not fit into the viewing window, the path will be compressed. To view the full path to the file, check the Full Path option.</p>
Save to File	<p>Button that lets you save information on the document's external links shown in the page window to a text file.</p>
Tab Attributes	<p>This tab appears when viewing the information about a graphics document or a BOM. It contains a list of attribute types assigned to the current document.</p>

* This field is filled in automatically and cannot be edited by the user.

By default, the author and company information match those specified when installing KOMPAS-3D (in the **User Information** dialog). This information, including comments, may also be obtained from document properties (see Section 5.1.4.1.1 on p. 1385).

If necessary, you can modify the default data (see Section 9.2.3 on p. 1748).

1.4. General Document

1.4.1. Using the Document Viewing

1.4.1.1. Graphic Cursor

The cursor is the main working tool in KOMPAS-3D. The cursor is used to execute commands from menus and buttons, create and edit objects, and perform numerous other actions.

The shape of the cursor depends on the active document type and the operation being performed.

The standard shape of the cursor located within the field of a graphic document or model document is a square “trap”. The cursor parameters (size, color, etc.) can be customized. This customization is made in the dialog activated by the **Service — Settings... — System — Graphic Editor — Cursor** command.

The following methods on cursor controlling are possible.

- ▼ **Moving using the mouse** is the main method that is available in all types of documents.
- ▼ **Moving using the keyboard.** You can move the cursor by using the arrow keys on the main or extended keyboard. In this case, the movement will not be arbitrary as in the case of using the mouse, but rather incremental.

When within a text document and BOM (while entering the text portion of the object), pressing an arrow button moves the cursor by one character or one line.

The minimal increment of the cursor motion in graphical documents, as governed by clicking an arrow button, obeys the defined cursor step. The cursor step value can be specified in the **Cursor's Current Step** on the **Current Status** bar. The step value can be entered from the keyboard or selected from the list. To quickly activate the **Cursor's Current Step** field, use the keyboard shortcut *<Shift> + </>* (press the *</>* key on the numerical keypad). The list of steps and default step can be specified in the cursor setup dialog (see Section 9.1.7.1 on p. 1703).

- ▼ **Setting Cursor Coordinates.** When working with graphic documents, you can enter the coordinates of the point where the cursor should be placed in the **Cursor Coordinates** on the **Current Status** bar.

To quickly activate these fields, use the keyboard shortcut *<Alt> + <X>*. After pressing it, the X coordinate field becomes active. Enter the desired value there. To go to the Y coordinate field, press the *<Tab>* key. Enter the value.

Confirm the new cursor coordinates value by pressing the *<Enter>* key.

You can specify the increments of the current coordinates in the cursor coordinate fields. To do this, enter the *^* character, and then — the increment value



If the **Current Status** bar is arranged vertically, click the relevant buttons to access the **Cursor's Current Step** or **Cursor's Coordinates** fields.

- ▼ **Fixing the cursor.** In graphic documents and model documents, after you set the cursor in the desired point you need to **fix** it — confirm that this exact point should be used for creating the object.

You can fix the cursor by left-clicking the mouse or pressing *<Enter>*.

For example, to create a line you need to specify and fix its start and end points. When creating a hole in a model, you can change its default location, specifying and fixing a new location of the anchor point.

1.4.1.2. Document view scale

KOMPAS-3D offers a wide selection of tools for zooming the image. It must be understood that changing the image scale does not affect the actual size of the objects (part dimensions, line length, font size, etc.). To draw an analogy: a sheet with a drawing is attached to a drawing board, and you are moving relative to it. When you move to or from the drawing, you will see a bigger or smaller image.

The zoom commands are grouped under the **View** menu. The main ones can be called by the buttons on the **View** bar.

The set of available commands for zooming the image depends on the current document type.

The commands for zooming the image in or out and specifying the zoom factor are available at all times.

The automatic scale setting is available when you work with text documents or BOMs, enter BOM text, and in some other modes.

The remaining zoom change commands are available in graphic documents and model documents.

1.4.1.2.1. Zooming Image In or Out



To zoom the image in or out, click **Zoom In** or **Zoom Out**.

By default, the zoom factor is 1.2.



In order to change the factor for model documents, call the **Service — Parameters...** command. — **System — Model Editor — View Control** command.

In order to change the factor for graphic documents, call the **Service — Parameters...** command. — **System— Graphic Editor— Edit** command.

In order the change this factor for text documents, reports and BOMs, call the **Service — Parameters...** command — **System— Text Editor— Edit** command.

In the right-hand part of the appearing dialog, type in or select from the list the desired value of the **Scale Change Factor**.



By default, when you work with graphic documents and models, the buttons for zoom in or zoom out are not available on the **View** toolbar. You can add these buttons to the toolbar (see section 9.4.2.1 on p. 1872).

1.4.1.2.2. Explicit Setting of Image Scale

The view scale in the current window is shown in the **Current Zoom** field on the **View** bar.



To change the scale, unfold the list and select the desired value. You can also type the value on the keyboard.



If the bar is positioned vertically, click the **Current Zoom** button to access this field.

If you need to set the current scale factor to 1.0, use the **Scale 1.0** command.

1.4.1.2.3. Zooming in a Random Image Section



To zoom in a random image section, click **Zoom In by Marquee**.

Click with the mouse on the first corner location of the zoom area. Then move the cursor to get the desired marquee size. The ghost marquee box will be reflected on the screen.

After fixing the second corner of the box, the image will be magnified so that the document portion within the frame will fit in the entire window.

1.4.1.2.4. Zoom to Selection



You can set the maximum view scale so that all objects selected in the document will fully fit in the window (fit the scale to the size of the selected group of objects).

To do this: click **Zoom to Selection**.

Selected construction lines and empty (containing no objects) views are ignored when running the command. A single selected point is also disregarded.

The **Zoom to selection** command is inaccessible if no object has been selected (see Section 3.1.3 on p. 876 for techniques of selecting objects).

1.4.1.2.5. Smooth Zooming



If the required zoom factor is unknown, you can select it visually by zooming the image in or out.



To do this: click **Zoom In/Out**.

The external cursor appearance will change.

Press the left mouse button and move the cursor vertically. As the cursor moves up, the image will be smoothly zoomed in, in the reverse direction — zoomed out. The center of scaling is the point, in which the left mouse button was pressed. When you reach the end of the screen, release the mouse button, move the cursor to the middle of the screen and then press again the left mouse button and move the mouse.

To have the panning center in the center of the screen, hold the *<Shift>* key pressed.

If you are using a mouse with a wheel, then you can spin the mouse wheel to pan the image. The panning center is at the point, where you start moving the wheel. You can also adjust the direction of the mouse's wheel rotation (see Section 9.1.2.10 on p. 1678).

To have the panning center in the center of the screen, hold the *<Shift>* key pressed.

The mouse wheel can be also used for panning an image in text documents and BOM documents. To do this, rotate the wheel and keep pressed button *<Ctrl>*.

1.4.1.2.6. Display the whole sketch



You can set the maximum view scale of the current sketch so that it is fully fit in the model window. To do this, call the **Show the sketch in full** command.

Construction lines and projections, created in a sketch are ignored when running the command.

1.4.1.2.7. Displaying Entire Document



To display the entire document being edited, call the **Show All** command.

1.4.1.2.8. Transition to the Previous or Next Display Scale

When working with graphic documents or models, you can return to one of the previous display scales in the window.



To return to the previous scale, call the **Previous Zoom** command.



To advance to the next scale, call the **Next Zoom** command.

1.4.1.2.9. Auto Scale

You can change the scale of the current text document, specifications, report, title block table or BOM so that the document page will fully fit in the window horizontally (the horizontal size of the image becomes equal to the window width) or vertically (the vertical size of the image becomes equal to the window height).



For the scale to be set automatically, call the **Zoom to Page Height** or **Zoom to Page Width** command.



1.4.1.3. Moving the Image

Moving (scrolling) the image means moving the document in the window without changing the image scale.

It must be understood that image offset in the window not affect the actual objects position in the document. To draw an analogy: a sheet with a drawing is attached to a drawing board, and you are moving relative to it. As you move to the left, to the right, up or down, your field of vision will include not the entire image but only a part of it.



To move an image while working with a graphic document or model, call the **View — Move** command or press the **Move** button of the **View** toolbar.



The cursor will change its shape.

The Move command can be actuated by pressing the keyboard shortcut **<Shift>+<Ctrl>** and then the left mouse button.

Move the cursor while holding the mouse button pressed (if you have actuated the command using the **<Shift>+<Ctrl>** keyboard shortcut, hold these keys as well). When you reach the end of the screen, release the mouse button, move the cursor to the middle of the screen and then press again the left mouse button and move the mouse.

Once the working field is moved into the desired position, release the mouse button and the keys.

If you use a mouse with a wheel or a three-button mouse, then to move the image in a graphic, text or BOM document, move the mouse and hold the middle button or wheel pressed.

Basically, this image moving technique can be used to increase the size of the document working field. This technique is convenient to use when the range of the scroll bars is not enough.

1.4.1.3.1. Scroll Bars

The scroll bars allow moving the image either vertically or horizontally. They are displayed in the document window at the bottom and on the right.

To scroll the image in the window, keep pressing the arrow buttons located at the ends of the rulers. You can also move the slider on the scroll bar.

If you are using a mouse with a wheel, then to scroll the image while working with text documents and BOM documents you can spin the mouse wheel. To scroll page by page, you need to hold the <Shift> key in addition.

You can configure the way the scroll bars are displayed in the current window. To do this, call the **Service — Parameters... — Current window — Scroll bars**.

The **Scroll Bars** group, which controls the way the scroll bars are displayed, is located on the right-hand side of the dialog that appears.

1.4.1.4. Refresh Image

While running various creation and editing commands, auxiliary lines and symbols may be appearing on the screen. In most cases, KOMPAS-3D automatically deletes those temporary objects upon completing a command. However, sometimes you may need to force the screen cleanup from “debris” — refresh the image.



To refresh the image in the active window, click **Refresh Screen**.

The refreshing also allows re-displaying objects with corrupted images. Such corruption is caused by removing or moving one of the superimposed or overlapping objects. For example, when you delete construction lines from the screen, the drawn outline can also disappear. It will reappear after you refresh the image.

1.4.1.5. Leafing through Documents

When working with text documents or BOMs, as well as when entering BOM text, you can leaf through the document — scroll it page by page.

To move sequentially from one page to the next, press <Page Up> and <Page Down>. To go to a specific page enter its number in the **Current page** field on the **Current status** bar and press <Enter>. The cursor will be set at the start of the first line on the page with the specified number.



If the **Current status** bar is positioned vertically, then to access the **Current page** field you need to click the respective button.

1.4.1.6. Undoing and Redoing Actions

Actions can be undone and redone using the **Undo** and **Redo** commands in the **Edit** menu. The respective command buttons are located on the **Standard** panel.



Command **Undo** reverts the document to a state it was in before the last action.

Redo, repeats the reverted action. The redo command is available only after you use the undo command.



Not all commands can be undone and redone. In particular, this applies to the commands of filling the title block, removing the entire document contents, saving the document to disk.



Note that all parameter editing operations (for example, changing unit sizes) for a component inserted into the assembly from the library are considered a single operation. Thus, it is impossible to cancel one of the library component editing operation: the **Cancel** command deletes the component from the assembly, while the **Repeat** command returns it with the parameters it had prior to the deletion.

In a graphic document, besides undoing and redoing commands, you can undo and redo actions within the current command. In this case, only those actions that pertain to the given command are undone. For example, you have constructed several circles in your document. If, without ending the **Circle** command, you click **Undo**, the created circles will disappear one by one, and after you click the **Redo** button, they will be restored.

You can configure the parameters of the undoing process for graphic and text documents: set up number of steps and also enable or disable grouping of similar operations (see Sections 9.1.7.11 on p. 1713 and 9.1.8.2 on p. 1721).

1.4.1.7. Recalling Recent Commands

The last command can be recalled not only from the toolbar, but also from the context menu or the **Edit** menu. This can be accomplished by clicking **Recall: (Command)**. You can execute a command by pressing the <F4> key. Besides the last executed command, several recent commands are available in the context menu (Fig. 1.4.1).

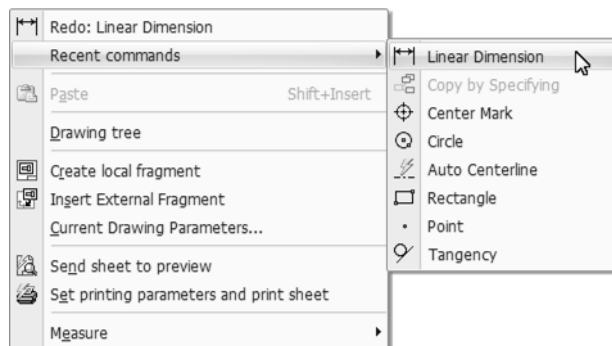


Fig. 1.4.1. Example of the recent commands list while working with a drawing
Recent commands lists are created for the following KOMPAS-3D document types:

- ▼ part
- ▼ assembly
- ▼ drawing
- ▼ fragment

- ▼ BOM
as well as for the following operating modes:
- ▼ sketch mode,
- ▼ table editing mode.

When you work with a text editor, you can recall only one last command. The recall command is executed in the **Edit** menu.

Categories of commands added to lists are shown in Table 1.4.1.

Tab. 1.4.1. Categories of commands that can be recalled depending on the document type and operating mode

Document type or operating mode name	Command categories*
Part	Edit, Tools, Select, Edit Model, Features of Sheet Body, 3D Lines, Surfaces, Construction Geometry, Dimensions and Process Symbols, 3D Measurements, Filters, BOM
Assembly	Edit, Tools, Select, Edit Model, Mates, 3D Lines, Surfaces, Construction Geometry, Layout Elements, 3D Measurements, Filters, BOM
Drawing	Edit, Tools, Select, 2D Measurements, Views, Pages Management,
Fragment	Edit, Tools, Select, 2D Measurements, Views, Pages Management,
Bill of Materials (BOM)	All edit commands, all BOM commands, commands of libraries working with BOM.
Text Document	All Commands of Text Editing
Table editing mode	Edit, Insert to Text, all table editing commands
Sketch mode	Edit, Tools, Select, 2D Measurements

* Commands included in each category are listed in the **Interface Setup – Commands** dialog (see Fig. 9.4.1 on p. 1873).



The commands are included in the list regardless of whether they have been completed or not.

A command will not be recalled if it was executed by double-clicking an object.

If the command execution conditions have not been specified, it cannot be recalled. For example, to run the **Tracing** command, it is required to select the source objects. If no object has been selected in the document, the **Copy** command will not be available in the recent commands list.

The number of “memorized” commands can be set up in the dialog which is called by the **Service – Settings... command. – System – General – Recalling commands** command. In the same dialog, you can disable the displaying of the latest commands in the context menu. At the end of a KOMPAS-3D session, the command lists are emptied.

1.4.2. General Methods of Performing Operations

1.4.2.1. Properties Bar

The Property bar (Fig. 1.4.2) serves to control the process of running a command.

Toggle the Property bar display by the command **View – Toolbars– Property bar**.

Setting the appearance of the Properties bar is described in Section 9.1.3.6 on p. 1683.

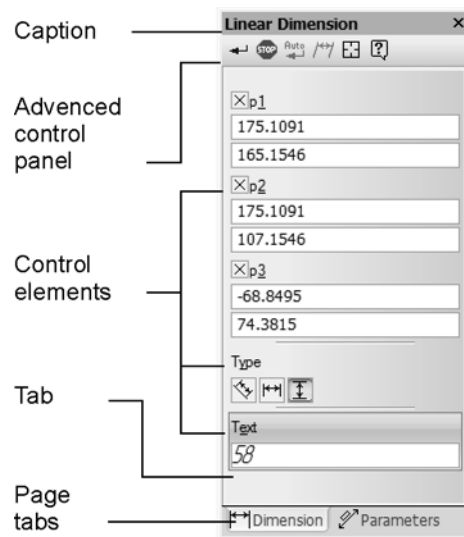


Fig. 1.4.2. Example of Property Bar

The Advanced Control Panel is located in the upper or in the left section of the Property Bar.

On the Advanced Control Panel there are the buttons, with which the special actions are executed, such as inputting an object, terminating the current command, turning On an auto object creation etc.

1.4.2.2. Property Window

The **Properties** window (Fig. 1.4.3) serves for viewing and changing the properties of the objects selected in a graphic document or in a model.

In graphical documents, the **Properties** window can be also used when copying properties between objects for viewing and selection of copied properties (see Section 3.4.1.2 on p. 1117).

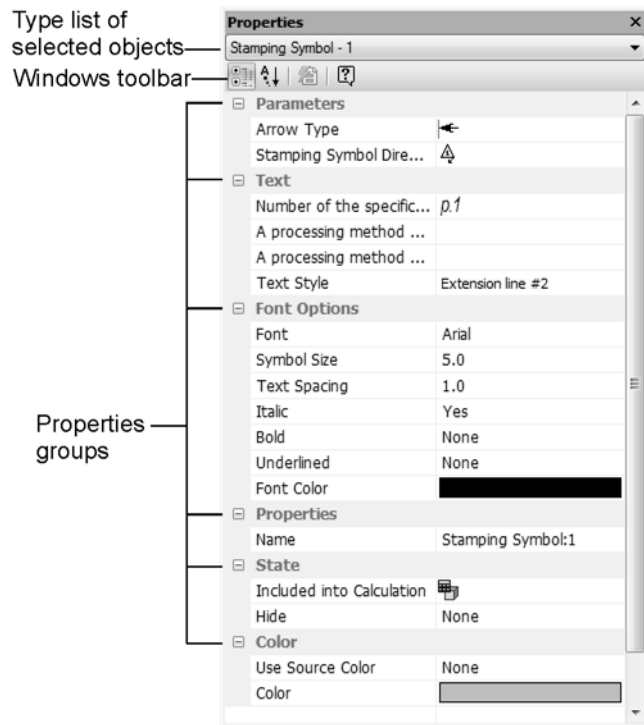


Fig. 1.4.3. Stamping symbol properties in a model



To display the **Properties** window on a screen, call the **Edit – Properties** command or press the **Properties** button in the Standard bar.



In a graphical document, the **Properties** command can be called from the context menu of selected objects.

The upper part of the **Properties** window contains a list of the types of the objects selected.

The window's toolbar can be found below the list.

The major part of the **Properties** window is occupied by a table of object properties.

The table consists of two columns. The first column contains the names of object properties, the second one contains the values of properties.



If the **Group** switch is enabled in the toolbar, the properties in the table are grouped. By default, groups are shown expanded, i.e. you can see on the sheet contents of groups— properties included in them. To minimize the group so as only its name would remain in a window, click on the “minus” icon to the left of the group name. If you click this icon again (after minimizing the group it is displayed as a “plus”), the group will be maximized.



If you enable the **Sorting** switch in the toolbar, the properties in the table will be sorted by names in alphabetical order.



The **Select by Properties** as a command button for selecting objects by a set of properties, excluding objects from the selected ones by a set of properties, and inverting the selection (for

details, see Section 3.1.3.3 on p. 879). The button is available in graphical documents.

Peculiarities of working with 3D objects in the **Properties** window are described in Section 2.14.3.6.1 on p. 793; description of working with graphical objects can be found in Section 3.4.1.2.1 on p. 1118.

1.4.2.2.1. Changing Properties



Using the **Properties** window, you can change the properties of a separate object or a group of objects without editing each of them separately. If the window is not displayed on the screen, enable it by clicking the **Properties** button.

To change the properties of objects, you must select them using any method. When editing the properties, you can add objects to the objects already selected or exclude any objects from those selected using any method.

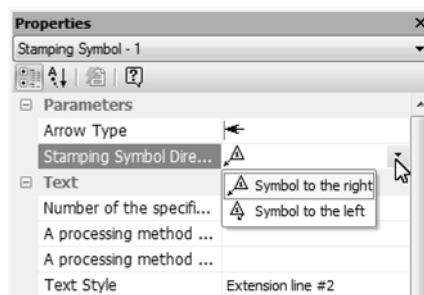


Fig. 1.4.4. Object properties changing

The list of object types in the **Properties** window contains the **All — N** line, where N is the total number of objects selected, and the lines are formed by the **Type Name — n** template, where Type Name is the name of object type and n is the number of the selected objects of this type (Fig. 1.4.4).



If there are selected the objects of one type only, then the **All** line does not present in the object type list.

If additional objects are selected or some objects are excluded from the selected ones, the list of types dynamically changes.

The table of properties represents:

- ▼ Names and values of all properties of selected objects of the same type— if a certain type of objects is chosen in the type list
- ▼ Names and values of the properties, which are common for all selected objects, if the **All** line is selected in the list of types.



Values of properties, that are not available for editing, are displayed in the cell in gray color.

To change the value of any property, perform the following operations:

1. Click the cell of the property value to be changed.
 - ▼ If the value is numerical (e.g. radius of circle), the cell will become available for its entry.
 - ▼ If the value is non-numerical (e.g. line style), the button with a black triangle appears in the right part of the cell. It makes it possible to open the list of values.
 - ▼ In case of a text value (e.g. surface finish parameter), the text input dialog will be displayed.
 - ▼ If the value is a hyperlink, the hyperlink editing dialog will be displayed.
2. Modify the property's value.
3. Press the *<Enter>* key or click outside the editable cell to confirm input of a numerical value. Close the text input or hyperlink editing dialog with the **OK** button. The non-numeric values do not require confirmation.

The objects (object) will be modified automatically in accordance with the new property value. In this case, the objects remain selected, and you can continue to change their properties.

Peculiarities of working with the **Properties** window in graphical documents are described in Section 3.4.1.2 on p. 1117; description of working with the window in models can be found in Section 2.14.3.6.1 on p. 793.

1.4.2.3. Running and stopping operations

To run an operation, you should call its respective command. For this purpose, you can choose the command name in the menu or click the respective command's button on the toolbar. For some commands, calling is possible only from menu or only from toolbar.

Once an operation has been run, the elements appear on the toolbar to control the parameters of the operation and/or object to be created in the document as a result of the operation.



To run the majority of operations, confirmation is required. To confirm operation, click the **Create Object** button on the Special Control panel or press the *<Ctrl> + <Enter>* key combination. The **Create Object** command is available in the context menu and in the **Editor** menu.

For some operations, objects can be created automatically. By default, Auto-sorting is enabled. In this case, operation is confirmed automatically right after setting the required minimum of parameters. For example, a circle is automatically fixed in graphic document if two points are defined: center and point on circle. For more information on automatic and manual creation of objects refer to Section 1.4.2.10 on p. 93.



Some operations allow to create several object of the same type in a row (or to perform several actions of the same type). For example, several segments can be built by running one segment building operation. To exit the operation after construction of the last object, click the **Interrupt command** button on the Special Control bar.

It is impossible to create several objects one by one (to perform several actions) for other operations. Such operations are automatically finalized after confirmation. For example, only one sector can be cleared using the respective operation. Pressing the **Interrupt Command** button before confirmation on operation performing means canceling an object creation (an action completing).

The **Interrupt Command** command is also available under the context menu.

To finish operation you can also perform one of the following actions:

- ▼ Press the <Esc> button,
- ▼ release the command button,
- ▼ execute any other command.

1.4.2.4. Object Parameters

After executing most of the object creation commands, you need to specify various parameters for these objects.

For example, after executing the command to create a circle, you have to specify the location of its center and its radius, and after executing the command to create an extruded solid — the extrusion direction, depth and draft value.

To create an object, it means to define all its parameters. When developing models and drawings using KOMPAS-3D, all parameters of the created objects are displayed on the Properties bar (see Section 1.2.2.1 on p. 52). Each parameter has a corresponding element on the bar (Fig. 1.4.5).

The parameters can be divided into numerical (point coordinates, length, angle, number of vertices, etc.) and non-numerical (line style, presence of axes of symmetry, etc.).

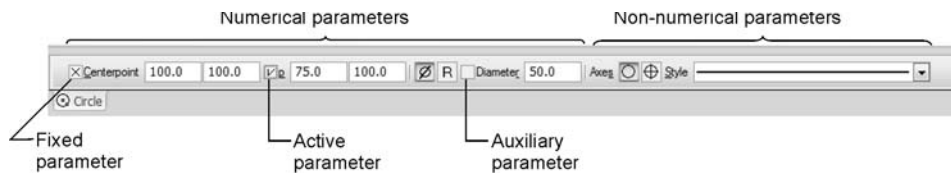





Fig. 1.4.5. Circle Parameter

A switch is provided next to the labels of most of the numerical parameters on the Property bar, which has an indicator corresponding to the parameter state (Table 1.4.2).

Tab. 1.4.2. Parameter status icons

	Name	Description
	Fixed	The box of a fixed parameter has a cross. This parameter value is accepted by the system. It will stay unchanged through the modifications of the other parameters and will be displayed on the object ghost. Any numerical parameters can be fixed.
	Active	A “check mark” is displayed on the active parameter’s box. The system expects this parameter value to be entered by selecting a point with the mouse in the document window. Active can be only the parameters representing the point coordinates.

Tab. 1.4.2. Parameter status icons

	Name	Description
	Auxiliary	The auxiliary parameter switch is empty. This parameter value is either not yet defined or depends on other parameter values (in this case, it shall be recorded automatically upon recording the parameter, on which it depends). An auxiliary parameter can be defined and fixed at any time. Before fixing, the auxiliary parameter value stays in the field. Any numerical parameters can serve as auxiliary parameters.

Unlike the non-numerical object parameters, which allow definition solely by selecting the desired choice in the Property bar, there are several ways to define numerical parameters.

- ▼ The first, most simple and obvious method of defining parameters is selecting the desired points in the document window (see Section 1.4.2.4.1 on p. 88). This method can be applied mostly to graphical objects, since among the three-dimensional objects there are too few whose parameters can be defined by specifying just points.
- ▼ The second method — entering parameters in a predefined order — allows more flexible control over the object parameters (see Section 1.4.2.4.2 on p. 88). This method is available when creating most of the objects — both graphical and three-dimensional.
- ▼ The third method is defining parameter values on the Property bar — which is less visual but universal and can be used when creating objects of any type (see Section 1.4.2.5 on p. 90).

You can combine any of the described methods of defining object parameters.

After all object parameters are defined, you need to confirm its creation. This can be done in one of the following ways:



- ▼ click the **Create Object** button on the Special Control panel,
- ▼ call the **Create Object** command from the **Editor** menu or from the context menu,
- ▼ press the keyboard shortcut `<Ctrl> + <Enter>`.

An automatic creation capability is provided in most commands for creating graphical objects (see Section 1.4.2.10 on p. 93). By default, Auto-sorting is enabled.

In subsequent descriptions of the object creation commands, the word “click” means pointing with the cursor to a specific element on the working field (i.e., placing the cursor on top of this object and left-clicking), while the word “specify” means specifying parameters in any way.



The chapters devoted to object creation describe the easiest and most commonly used ways. This applies to both the way and procedure of specifying the object properties. For example, to create a line, you have to specify its length, direction and position. You can do it like this, for example: enter the start point coordinates, then the slope angle to the X axis of the current coordinate system, and, finally, click the end point of the line with the mouse. Naturally, this way is as correct as all the others; moreover, in some cases it can be the only possible way. Still, the description of the line segment creation contains only two basic creation techniques: using two points and using one point, length and angle, since these two techniques are the easiest and most commonly used.

1.4.2.4.1. Specifying Points in the Document Window

To specify a point in the document window, it means to set the mouse pointer in it and fix.

In a graphic document and in a sketch, you can use snaps to position the cursor accurately (see Section 3.1.1 on p. 865) or the cursor coordinate fields on the **Current status** bar (Fig. 1.4.6).

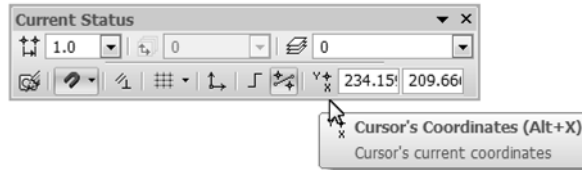


Fig. 1.4.6. Cursor coordinate fields

Fixing the point at the cursor location is necessary to have this point coordinates passed to the Property bar fields (i.e., to make them the object parameters). A point can be fixed in two ways:

- ▼ by clicking the left mouse button (this fixing method is convenient if the point was selected by the mouse “by sight” or with the use of a snap),
- ▼ by pressing the `<Space>` or `<Enter>` key (this is convenient if the point is specified with the use of the keyboard — by typing in the cursor coordinates or by using keyboard snapping).

For example, to create a line segment, it is enough to specify and fix two points: the start and the end. Note that in this case not only the line endpoint coordinates will be entered in the Property bar fields, but also its length and angle (they are calculated automatically).

You can change the default sequence of specifying the points, i.e. specify the parameters in a sequence other than that offered by the system (see Section 1.4.2.8 on p. 91).

1.4.2.4.2. Entering Parameters in a Predefined Order

This method of defining object parameters implies the following.

The order of entering parameters, which are not point coordinates (length, angle, distance, name, etc.), is predefined for various objects and is stored in the system. Therefore, the value (number or text), typed in from the keyboard at the time of creating or editing an object, is instantly accepted by the system as the value of a predefined parameter and is entered in the predefined field. To cancel an entered value, press the `<Esc>` key; to fix it and proceed to the next predefined field — press `<Enter>`. When specifying a point or object in the document window, the fixing of the input value and jumping to the next parameter occur automatically.

The parameters of the predefined input can be assigned in an arbitrary order. To navigate through the predefined fields in the forward direction, use the `<Tab>` key, in the reverse direction - the `<Shift>+<Tab>` keys. Upon fixing the value in the selected field, there is an automatic jump to the next field of the predefined input.

If the currently focused field is **Text** (for example, when creating a Surface Finish symbol), then the respective dialog appears on the screen upon entering the first characters of the text.

Let's review in detail the method of inputting parameters in the predefined order on the example of creating a line segment, for which we know the start point, the length and the inclination angle. Obviously, constructing it by specifying the start and end points is not rational in the least.

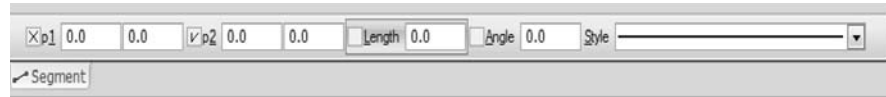


Fig. 1.4.7. Property bar when creating a line segment

The predefined sequence of inputting line parameters includes the **Length** and **Angle** fields. Upon calling the **Line** command, the **Length** field is focused on in the Properties bar — it is framed and has a different background (see Fig. 1.4.7). Enter the line length. You will see this value entered in the **Length** field. Note that the field is de-focused. This happened because the Properties bar became active when you were entering a value, and the predefined entry mode is active only when the document window is active (for details on switching between the Properties bar and the document window see the following section). Specify by the mouse the start point of the line in the window. You will see a ghost line of the specified length with the first point fixed in the document window. Since the document window became activated upon selecting the point, the predefined input mode is back On: the next field has become focused on in the Properties bar — **Angle**. Enter the angle value. It will be entered in the **Angle** field. Press **<Enter>**. The angle value and the line itself will be fixed.



You do not have to specify the start point of a line after entering the length. This can be done either before defining the length, or after defining the angle.

If for some reason you need to specify the line's inclination angle first, and then its length, do the following.

1. Call the **Line** command. Without exiting the predefined entry mode, i.e., without entering the length value, press **<Tab>**.

The focus will switch from the **Length** field to the next predefined field — the **Angle**.



Since the Properties bar activates upon inputting the value of the predefined parameter, then pressing the **<Tab>** key or the **<Shift>+<Tab>** key combination activates at this moment the next or the previous control on the Properties bar.

2. Enter a new value and press the **<Enter>** key.
The angle will be fixed, and the focus will automatically switch to the **Length** field.
3. Enter a new value and press the **<Enter>** key.
The line will be fixed.

Switching between the Properties Bar and the Document Window

The document window becomes active immediately after you execute the command to create a graphic or 3D object, meaning that the system waits for you to specify the object parameters by clicking points or existing objects.

The part's Properties bar becomes active at the time of switching to defining the object parameters by the Properties bar. This happens when:

- ▼ entering values in the predefined fields (see the previous section),

- ▼ activating the Properties bar control by the mouse or an accelerator key,
- ▼ clicking the mouse on a free space on the tab,
- ▼ switching between tabs by clicking on a tab tag.

Note that the Properties bar does not activate when clicking its title, using its context menu, clicking any other button on the Advanced Control panel or using the geometrical calculator.

When the Properties bar is active, switching to the document window occurs when:

- ▼ the *<Enter>* or the *<Esc>* key is pressed,
- ▼ clicking the mouse in the window,
- ▼ switching to another Windows application window.

1.4.2.5. Entering Values in the Properties Bar Fields

To explicitly define a parameter value in a field of the Properties bar, click the left mouse button in that field. It will become editable. Enter the desired number.

Another way to access a parameter field is pressing the *<Alt>* key and another key corresponding to the underlined letter in the parameter name (for example, *<Alt>+<E>* to enter the line inclination angle).

The numerical parameter values of the graphical objects can be “read” from the existing objects by means of the geometric calculator (see Section 3.1.2 on p. 871). Note that calling the geometric calculator on a parameter field is possible if this field is **not active**.

In the Properties bar fields you can enter not only the numerical values of the parameters but also the expressions for their calculation. Syntax of expressions is described in detail in Section 7.1.3.3.1 on p. 1580.



If an expression begins with the *<^>* symbol, for example, *^5+M_PI*, the expression calculation result will be added to the value contained in the field prior to entering the expression.

After entering an expression, press *<Enter>* to calculate it.

The described methods can be used for parameters in different states: fixed, active, auxiliary (see Fig. 1.4.5 and Table. 1.4.2 on p. 86).

When jumping to the next parameter, the value of the current parameter is automatically fixed.

The numerical values in the Properties bar fields are displayed with the precision defined in the ‘Number Scheme Setup’ dialog (see Section 9.1.2.2 on p. 1668). This precision does not affect the parameter value stored inside the system — it is always equal to the number specified by the user.



Setting the cursor in a field with an already defined parameter and pressing the *<Enter>* key means a repeated input of this parameter value. At the same time, the system stores the value that was displayed at the moment you pressed *<Enter>*, meaning that the number is stored with the number of decimal places displayed.

1.4.2.6. Fixing Parameters

Fixation is possible for all the numerical parameter values of the graphical objects and the point coordinates of 3D objects.

Value of fixed parameter stays constant, other parameters become available for modification. The indication of a fixed parameter is a cross displayed on the switch next to the parameter field (see Fig.1.4.5 and Table 1.4.2 on p. 86).

Parameter fixing can be viewed as limitation of the possible configurations of the object being created.

Suppose you need to draw a line whose length should be exactly 60 mm. Fix the line start point, and then enter the value 60 in the line length field and press the *<Enter>* key. Now, upon any cursor movements, only the line inclination angle will change, while the length will remain equal to 60.

Notably, when you specify a point on the working field or remove a parameter of an existing object (see Section 3.1.2 on p. 871), this parameter becomes fixed automatically.

If, while moving the cursor, you reach a desired value of any parameter and want to fix it, without moving the mouse activate the parameter field using the relevant keyboard shortcut (for example, *<Alt>+<E>* for the line slope angle) and then press *<Enter>*.

1.4.2.7. Releasing Parameters

To cancel the fixation of a parameter value, left-click the switch next to the parameter name. The fact that the fixation has been canceled is indicated by the absence of a cross on the relevant switch (see Fig. 1.4.5 and Table 1.4.2 on p. 86).

Assume that while drawing a line you specified the start point and then entered and fixed the length value equal to 60 millimeters. After that, upon any cursor movement, only the line inclination angle changed. If you now release the line length, then you will be able again to vary either the inclination angle or the length.

While creating an object, you can repeatedly fix or release its parameters.

1.4.2.8. Activating Parameters



This method is accessible in graphic documents when constructing the following objects: arbitrary, parallel and perpendicular line segments, circles by two points, by three points, tangent line to one curve, arcs.

Parameters defined by fixing the cursor at a certain point within the document (for example, point coordinates, start and end angles of an arc) may be activated.

When defining such parameter, a “check mark” is displayed on the parameter’s switch, meaning that the parameter is active (see Fig. 1.4.5 and Table 1.4.2 on p. 86), i.e., the system expects a value to be input for this particular parameter. After you specify the value, a parameter becomes fixed, and the next parameter becomes active.

When creating objects, their parameters become active in the order set in the system by default. However, the user can change it if necessary.

To activate an object parameter, click the left mouse button on its name (not on the switch, nor in the input field!). The parameter will become active, and you will be able to specify its value.

Suppose, you need to create a line with a specified length, parallel to another one and ending at a given point. Call the parallel line creation command, select the current line, enter the line length and fix it. By default, the point that you select in the document field will be treated as the start point of the line, therefore currently “checked” is the **p1** parameter. To be able to select that same point as the end point of the line, activate the **p2** parameter by clicking the mouse on its name. The check mark on the switch of parameter **p1** will disappear, appearing on the switch of parameter **p2**; the ghost of the line being entered will rotate by 180°. Now specify the line end point, for example, executing the snapping to the required point.

You can activate both the auxiliary and fixed parameters (for example, when editing objects). In this case, the activation of a parameter is identical to releasing its value.

1.4.2.9. Saving Parameters

Quite often you need to draw several objects with a number of identical parameters. A typical example is concentric circles (their identical parameter is the center point coordinates).

KOMPAS-3D allows memorizing parameter values and using them before the **current command is completed** when creating the next objects.

Perform the following actions to use identical parameters while creating objects.

1. Specify the parameters that have to be memorized.
2. Click the **Remember Status** button or press the `<Ctrl> + <Shift> + <P>` key combination. The button will stay pressed, indicating that the parameter is remembered.
3. Keep constructing as long as the remembered parameters are needed.
4. Click the **Remember Status** button.



If the entered parameters uniquely define the object (for example, an already fixed center point and circle radius), then the **Remember Status** button will become inaccessible.

You can memorize parameters when completing various operations. Below are several examples of how this functionality can be used. After reviewing them, try to create the following object groups:

- ▼ concentric arcs with an identical opening of the angle,
- ▼ same-length lines with the same start point,
- ▼ same-length lines parallel to the same object,
as well as perform the following actions:
- ▼ make identical the radii of several circles and arcs,
- ▼ measure the length of several sections of a curve starting at the same point.

Example 1. Concentric Circles

1. Click **Tools — Geometry — Circles — Circle**.
2. Specify the circle center point.
3. Click the **Remember Status** button.

4. Now subsequently select the points lying on the respective circles.



If the **Auto Create Object** button on the Special Control panel is pressed, the circles will be created automatically. Otherwise, you will have to confirm the creation of each circle by clicking the **Create Object** button.

The automatic and manual object creation is detailed in Section 1.4.2.10 on p. 93.

If you unfix the center coordinates and then select another center point, you can create a new family of concentric circles.

Example 2. Placement of Several Points on a Single Vertical

1. Call the command **Tools — Adding Constraints — Points — Align Handles Vertically**.
2. Select the first point for a symmetric constraint.
3. Click the **Remember Status** button.
4. Sequentially specify the remaining points to be aligned.



Example 3. Measuring the Distance from one Point to Several Others

1. Call the command **Service — Measure — Distance between 2 Points**.
2. Specify the point from which you want to measure the distance.
3. Click the **Remember Status** button.
4. Sequentially specify the points to which you want to measure the distance.

The values of the measured distances will appear in the info window.

1.4.2.10. Automatic and Manual Object Creation

When you modify object parameters while constructing an object, sometimes you will not need to create an object immediately after specifying all defining parameters. It is convenient to decide first whether the parameter values are correct, and only after that to confirm the object creation.

Two buttons are displayed after you execute the majority of the object entry commands on the special control panel (see Fig.1.2.2.1 on p. 52).



One of them, **Object Auto Creation**, is pressed by default. While it is in pressed state, all objects are created (fixed) immediately after you enter parameters sufficient for constructing it. The remaining parameters of the object will retain current values. For example, in order to create a circle, it is sufficient to set the center's position and a point on the circle. Therefore, if autocreation is enabled, the circle is fixed in the document after specification of two points. Presence of axes depends on the switch enabled in the **Axes** group. Thus, in order to create a circle with axes, ensure that axes drawing is enabled prior to center and point on the circle setting. If it is disabled, enable it.



If there is no need to create objects automatically, release the **Autocreation** button. Now, to confirm the creation of every subsequent object, you will additionally have to click the **Create Object** button. Until this button is pressed, the object is not fixed, so you can modify any object

parameter (see Section 1.4.2.8 on p. 91) any number of times. In the example above, it was possible to enable/disable circle axes both prior to and after specification of points. Each modification will be immediately reflected in the object ghost in the document window, helping you to make sure you are entering the correct values.

You can also call the **Autocreation** command (and, respectively, enable/disable autocreation of objects) using the `<Ctrl> + <Shift> + <Enter>` key combination.



The automatic creation is not provided for some graphic objects (for example, equidistant). You have to fix them manually.



To fix a graphic object, you can simply click it.



When describing commands for construction of objects for which auto create is supported, it is implied that it is enabled. The cases when autocreation should be disabled are specifically mentioned.

1.4.2.11. Reselecting Objects

When executing numerous commands, you are required to select an object that will be used as a reference object for construction.

Once you have selected an object, you can create several objects on its basis.



To select another object as the reference object and create a new group of objects, click the **Reselect** button or press the `<Ctrl> + <Shift> + <Z>` key combination. The constructed objects will be fixed, and the system will wait for you to select a reference object again.

You can use this button to construct several groups of lines parallel or perpendicular to a specific object, several groups of angular dimensions from a common datum, etc.

Another purpose of the **Reselect** button is reselecting the reference object without repeating the command.

For example, if you accidentally specified a wrong curve for constructing a tangent arc, click the **Reselect** button. The selected curve will be unselected, and the system will wait for you to specify another reference object.

1.4.2.12. Copying object properties

Sometimes, you may need a new or edited object has a number of properties similar to a previously created object. When working with graphical documents and sketches, it is possible to copy some properties of an existing object without leaving the creation or editing process.

The properties are copied according to the following rules:

- ▼ The **Current Line Style** property is copied among all objects allowing the change of a line style (for example, a circle line style can be copied into a line, equidistant line, wavy line, a brace, etc.).

- ▼ The properties **Text Style**, **Font Options** and **Paragraph Properties** are copied among all objects that contain text and allow changing of these properties.
- ▼ Some properties are copied only between objects of the same type (e.g. leader arrow view can be copied to another leader only and not to a dimension).

Object properties are copied using the **Copy Properties** command. The **Copy Properties** command is available in the Special Control Panel after calling most of the commands located in **Geometry**, **Dimensions**, **Designations**, and **Designations for Construction** bars.

In order to copy an object's properties:



1. In the Special Control Panel, click the **Copy Properties** button or select the **Copy Properties** command in the context menu.



The cursor will change its appearance.

2. Specify the object to be the source of properties being copied. For this purpose, hover the cursor over the object whose properties should be copied. When the selected object highlights, click the left mouse button.

The system will automatically return to the command from which the **Copy Properties** command has been called.

3. Specify the parameters of the new or edited object.
4. Finish creating or editing the object.

The object will have the properties of the source object.

The **Copy Properties** command can be interrupted before returning to the command from which it has been called using one of the following methods:



- ▼ Click the **Interrupt command** button the Special Control Panel,
- ▼ Call the **Interrupt command** command from the context menu.
- ▼ Press the <Esc> key.

The **Copy Properties** command can also be called using the <Ctrl>+<Shift>+<C> key combination.

1.4.2.13. Displaying Object Parameters Next to the Cursor

When you create and edit geometric objects, their parameters can be displayed not only in the Properties bar fields, but also next to the cursor (Fig.1.4.8) This helps control the correctness of the construction.



Fig. 1.4.8. Displaying object parameters next to the cursor:
a) when creating an arc using three points, b) when creating a line parallel to an existing line

The command parameters display is controlled by the **Display Commands Parameters** option (see Section 9.1.7.1 on p. 1703) in the cursor parameters setup dialog.

1.4.2.14. Rounding Parameter Values

When working with KOMPAS documents you can turn on the rounding of linear values. This function is used in various commands creating and editing graphical and three-dimensional objects.



The enabling and the switching off the Rounding mode in the current document window (windows) is done by the **Rounding** button on the **Current Status** bar or by the <F7> key.

The default setting of the rounding mode (On or Off) for new document windows is done in the Cursor Properties dialog (see Section 9.1.7.1 on p. 1703).

When On, the parameter values are rounded to the nearest value — a multiple of the cursor current step. The value is rounded to the smaller one if the difference between the value and the nearest multiple is less than half the cursor step; otherwise it is rounded to the greater one.

Let's review working in the rounding mode in detail based on a rectangle creation example. For this, do the following.



1. Make sure the rounding mode is turned off.
2. Set the cursor step equal to 5 mm.
3. Call the **Rectangle** command.
4. Specify the first rectangle's vertex position.
5. Move the cursor and watch the values in the **Height** and **Width** fields on the Properties bar.
6. You will see that these fields display decimal values with an accuracy of up to four decimal places.
7. Click the **Rounding** button.
8. Make sure the **Height** and **Width** fields now display integer values multiple of 5, i.e., the rectangle dimensions change in increments with the step equal to the cursor step.
9. Change the cursor step from 5 to 2.
10. Make sure the rectangle's dimensions became a multiple of 2.



Note that the rounding mode is temporarily suspended when a snap activates. For example, if the end point of a line segment being constructed is specified by some kind of a snap, say, **Nearest Point**, then the line length will not be rounded. It will be exactly equal to the distance from the start point to the point of the snap.

Additionally, no rounding of the fixed parameter values is done. For example, when constructing a line segment parallel to a given one, you can enter any value in the **Distance** field on the Properties bar and fix it. When selecting the start and end point of a line, its length will be rounded, while the distance to a reference will not.

The rounding of values when working with three-dimensional objects takes place when “dragging” their defining points. Note that in parts and assemblies it is possible to round not only linear but also angular values.

2. Three-dimensional modeling

2.1. General information

2.1.1. 3D Modeling Main Definitions

2.1.1.1. Model in KOMPAS-3D

KOMPAS-3D supports creating two **types of models**: part and assembly.

- **A Part** is a model type designated for presentation of products manufactured without assembly operations. Is created and stored in a document «Part», a file extension— *m3d*.
- **An Assembly** is a model type designated for presentation of products manufactured with assembly operations. Is created and stored in a document «Assembly», a file extension— *a3d*. Technological assembly is a variety of an assembly. Is created and stored in a “technological assembly” document: file extension — *a3d*.

Three-dimensional model in KOMPAS-3D consists of **objects**. Objects are divided into:

- geometrical,
- layout elements,
- measurement objects,
- components.

Geometrical objects include: bodies, surfaces, curves, points, sketches, construction geometry objects. Layout elements include dimensions, cosmetic thread, leader lines, surface finish symbols, datum symbols, positions, position reference numbers, shape and position tolerance symbols.

A Component is an object of a model which, in its turn, is a model: part or assembly.

Model objects are created and edited by performing **operations**. During object creating and editing its associative linking with other object is possible. An **associative relation** is a one-way dependence of an object's position or geometry on another object's position or geometry. The dependent object is considered **derivative**, while the object on which the derivative object depends, is considered to be a **source object** with respect to the derivative one.

You can assign MP calculation parameters to a model as a whole, or its parts (bodies, components)—material and material density, as well as you can assign **properties**—data about the product that the model (part of the model) represents.

Model contents, sequence of its construction, and relations between model objects are displayed in the **Design Tree**.

2.1.1.2. Model objects

2.1.1.2.1. Geometrical Objects

Geometrical objects consist of **primitives**. Primitives are:

- vertex,
- an edge,
- face.

Vertex— primitive, being a kind of a point or end of edge. Special vertex case is a zero length edge (for example, cone vertex)

An edge is a primitive representing a section of a curve or border line of a face limited by vertices and containing no other vertices inside. In special cases, an edge may not be limited by vertices (closed edges).

A face is a primitive representing a surface's part or a surface limited by edges and containing no other edges in it. In special cases, face may not be limited by edges (for example, spherical and toroidal faces).

Such objects like planes and axes do not have primitives.

Other objects, depending on their type, consist of one or several primitives. For example, object “point” consists of one vertex, polylines and sketches— of edges and vertices, bodies— of edges, vertices and faces.

Body is a model object having a certain volume and related to a material. A body does not have a separate file that represents it.

As a rule, body is a set of faces, edges and vertices. In special case, body may be represented as one face (for example, spherical or toroidal bodies).

Body faces form closed surface. The closeness failure results in the breach of the body's integrity.

Body construction in a model is described in Section 2.3.

Specific body type— **sheet bodies**. They are provided for modeling parts obtained from the sheet material using bending operations. Sheet body constructing is described in Section 2.4.

Surface is a geometric object represented by a coherent sum of faces or by one face. Faces of a surface cannot be faces of other objects (other surfaces and bodies).

Constructing surfaces is described in Section 2.6.

A sketch is a 3D model object created using the drawing graphical editor on a plane or on a flat face.

Sketches are used in some operations. For example, a sketch may shape a section of a body resulting from the extrusion operation, rib contour, etc.

Requirements to a sketch are defined by an operation in which it is used.

Drawing sketches is described in Section 2.2.

Construction geometry objects are:

- coordinate systems,

- coordinate and auxiliary planes,
- coordinate and auxiliary axes,
- control points,
- attachment points.

Constructing auxiliary geometric objects in a model is described in Section 2.8.

2.1.1.2.2. Layout Elements

Layout elements are:

- cosmetic thread,
- dimensions:
 - linear,
 - angular,
 - diameter,
 - radial,
- designations:
 - surface finish symbol,
 - Datum symbol,
 - leader
 - ref.No. leader,
 - shape and position tolerance symbol.

Constructing layout elements in a model is described in Section 2.9.

2.1.1.2.3. Measurement objects

Measurement object— an object containing measurement operation results.

Measurement objects are:

- distance and angle,
- edge length,
- area.

The measurement object value always corresponds to actual value of the model's measured parameter.

Making measurements in a model is described in Section 2.15.2.

2.1.1.2.4. Components

Component is an object of a model represented by another model saved in a separate file. Components include parts, assemblies, standard products and library elements. Local part is a special component type. A local part has no individual file representation and is stored within its model. Components, in turn, can consist of other components, etc.

One and the same component can be inserted in different models. You can insert one and the same component twice in a model.

Operations that imitate processing an assembled product, e.g. create a hole that passes through several components, can be performed within a component-containing model. The result of such operations is not passed to the component files.

Components can be linked by **matings**. Two types of matings are available:





- positioning matings fix an object with respect to another object in a certain way,
- mechanical matings define the law of how one object moves relative to another.

For details about adding components to a model and applying matings to them see Section 2.11.






2.1.1.3. Modes of working with a model

KOMPAS-3D offers special modes for working with 3D models. (Table 2.1.1). To enable and disable these modes, buttons of the **Modes** bar are used. Another exception is sketch mode and on-the-spot editing mode—their buttons are found on the **Current status** panel, and model section mode—its button is found on the **View** panel.

Tab. 2.1.1. Modes for working with models

	Name	Purpose	Specifics
	Sketch mode (see Section 2.2.1.2)	Creation of a new sketch or editing an existing sketch.	Commands for creation of geometric objects and dimensions are available.
	The mode for displaying a model in the perspective projection (see Section 2.1.3.5)	Displaying a model taking account of the perspective.	The degree of the introduced image perspective distortion can be adjusted.
	Surface smoothness test mode (see Section 2.15.4)	Visual assessment of smoothness of faces' joints, as well as identification of minor curvature changes inside the faces.	All faces of a model are presented as mirroring the specified image. The image's type can be customized.
	Model section mode (see Section 2.15.5)	Visual assessment of mutual position of parts and nodes of an assembly model, a part's design elements, its wall thickness, etc.	Model is displayed by a truncated intersecting plane or a set of planes. In operations in this mode you can specify vertices, edges, and faces of the model, including truncated, but you cannot specify vertices, edges, or faces of a section.

Tab. 2.1.1. Modes for working with models

	Name	Purpose	Specifics
	The mode for presentation of a sheet body in the fully fledged form (see Section 2.4.7.2)	Displaying bends, selected by the user, in the bent form, while other bends are displayed in the open form.	Only the following actions can be performed: <ul style="list-style-type: none"> – model viewing and printing; – modifying geometric and mass center characteristics of the model.
	Assembly component separation mode (see Section 2.15.7)	Displaying assembly component in the exploded form (this may be needed for a more intuitive appearance of the assembly).	Only the following actions can be performed: <ul style="list-style-type: none"> – model viewing and printing; – creation of reports.
	The mode for editing a component at place (see Section 2.11.6.2.1)	Editing the selected component in the current assembly window, surrounded by other components.	All construction and editing commands affect only the selected component. Other assembly components are shown in the window, but not accessible for editing, they can be used when executing commands (selecting faces, edges, vertices).
	The mode for recalculation of model's dimensions with tolerances applied (see Section 2.10.3)	Recalculation of model's dimensions taking account of the assigned tolerances. Dimensions of a model are calculated in accordance with the current recalculation (see Section 2.10.3.1).	Only the following actions can be performed: <ul style="list-style-type: none"> – model viewing and printing; – modifying geometric and mass center characteristics of the model. – creation of reports.
	The mode for displaying dimensions of a selected element (see Section 2.9.2.2.2)	<ul style="list-style-type: none"> – Viewing an object's (sketch or operation) dimensions outside the process of its editing. – Modifying object dimensions' values. 	Once an object is selected, its dimensions are displayed on the screen – dimensions of sketches and operations expressed in linear and angular values.

2.1.1.4. Base point of a three-dimensional object

Base point of a 3D object is a point to be used as the origin for construction of an object's geometry in a model. The base point position is determined by the system automatically and depends on the object's type and method of construction. For example, the base point of a component is the starting point of its absolute coordinate system while the base point of an operation built on a sketch is the center of gravity of the sketch curves.

The base point is represented on the screen as three mutually perpendicular vectors.

For example, the base point is displayed when a copied object for an array creation is selected (Fig. 2.1.1).

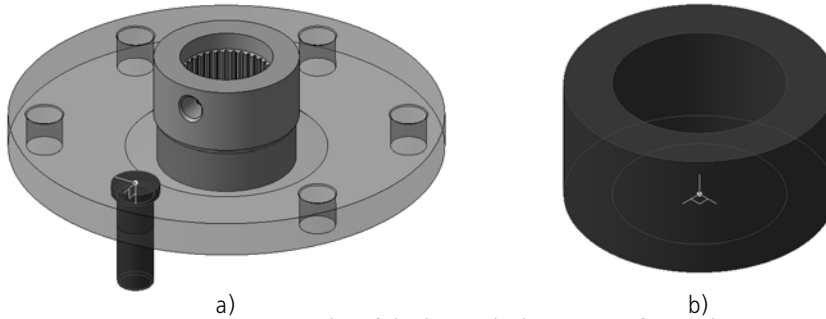


Fig. 2.1.1. Examples of displaying the base point of a 3D object:
a) a component's base point; b) base point of the extrusion operation built on a sketch

2.1.2. Model tree

During operations with any part or assembly, a window with a **Model Tree** may be displayed. This window is always positioned inside the model document window.

The window contains two tabs – **Construction** and **Versions**. The **Construction** tab displays the model's Design Tree. It is described in this section. The **Versions** tab displays the model's Tree of Versions. The tab is described in Section 2.13.1.2.





The model's Design Tree (hereafter – the Design Tree) is a graphical representation of an object set making up the model. Root object of the Design Tree is the model proper, i.e. a part or an assembly. Object icons automatically appear in the Design Tree right after such objects are created in the model. Section 2.1.2.5).

Model and assembly components may be located in the Tree in the order of their creation (see Section 2.1.2.1) or be combined by types and groups (see Section 2.1.2.2).

Components of a model—parts and subassemblies—are independent models. Therefore, their constituent objects are also placed in the relevant tree branches.

There is a control panel with four buttons in the top part of the **Construction** tab (Table 2.1.2).

Tab. 2.1.2. Buttons of the **Construction** tab toolbar

Name	Description
	Model structure display Controls method of data presentation in the Design Tree. If this button is enabled, the model structure is displayed in the tree window (see section 2.1.2.2); if the button is disabled, the tree window displays the model construction sequence (see section 2.1.2.1).
	Components of the Model tree Allows to specify which types of items will be displayed in the Design Tree, and which will be omitted. You can also enable component grouping for assemblies (see Section 2.1.2.3). Adjustment of structure is only enabled when the Model structure display button is selected.
	Relations Controls the special area display in which hierarchy of relations is shown for the object selected in the Design Tree (see Section 2.1.2.7.2). The area is displayed in the lower section of the Tree's window.
	Additional tree window Allows to create additional Design Tree window and to display the section or item which had been selected in the tree before pressing this button (see Section 2.1.2.4).

The Design Tree is intended not only for item fixing, it also facilitates selecting and specifying items during command execution (see Section 2.1.4.1.5).

Icon size in the Design Tree can be customized, see Section 9.1.3.8.7.

Context menus of items and sections of the Design Tree contain the most frequently used commands, such as: commands for item display control, commands for item inclusion in/exclusion from calculations, commands for editing, deletion etc.

The tree may contain additional information about the model represented as special icons used for marking objects. The list of such icons and their descriptions are given in table 2 Appendices I.



All commands of the section context menu, excluding the **Attributes** command, are applicable for all items included in the section.

The **Design Tree** may be hidden. To do this, call the **View – Model tree** command. To enable displaying of the window, call the command again. In the enabled state, a “checkmark” is displayed next to the command name in the menu.



The **View – Model tree** command controls only the main window display of the Tree. Additional Tree windows, if any, have to be closed manually.

If several windows are open with the same model document, then the **Design Tree** can be shown or hidden in any of them. Different sets of additional Tree windows may also exist in different model windows

2.1.2.1. Display of model design sequence in the Design Tree)



Stages of the model's construction are displayed in the Design Tree, if the **Display model's structure** button is disabled. The button is located in the toolbar of the **Construction** tab in the **Model Tree** window. Model objects are displayed in the Design Tree in the order of their creation (Fig. 2.1.2).



Fig. 2.1.2. Model creation sequence

The following features are accessible for operations with the Tree reflecting model creation sequence:

- Changing creation sequence (see Section 2.14.3.6.3),
- Use of creation finishing pointer (see section 2.1.2.6).

2.1.2.2. Display of model structure in the Design Tree



The model's structure is displayed in the Design Tree, if the **Display model's structure** button is disabled. The button is located in the toolbar of the **Construction** tab in the **Model Tree** window (Fig. 2.1.3).

The model's structure is a set of sections into which model's objects are grouped by types. These sections are displayed in the Design Tree. Each section of the Tree has a name and an icon.

A Design Tree section is a named part of the Tree containing model objects of the same type, for example, **Sketches**, **Bodies**, and so on.

Items inside sections are positioned in order of their creation. The section names, their icons and types of constituent objects are given in Table 1 of Appendix I.

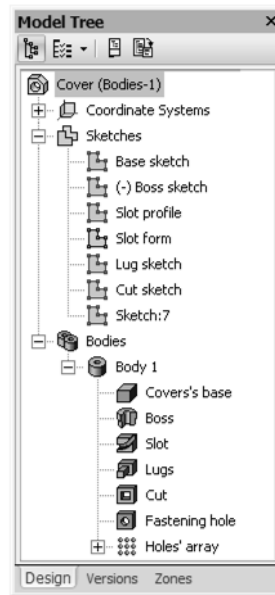


Fig. 2.1.3. Model structure

Such items as *Surface Splitting*, *Delete faces*, *Surface Knitting* may be located in section *Surface* or *Body* depending on what they refer to. For example, splitting operation of body surface is placed in the section related to this body, while surface splitting operation—in the *Surface* section.

The items which belong to several bodies, to a body and surfaces, or to a body and a component simultaneously, are located at level one of the Tree (i.e., a Chamfer object applied to body and surfaces; a Hole object applied to a body and to a component).

Additionally, some error operations are displayed at level one of the Design Tree (for instance, the operations containing an “Undefined Operation Applicability Range” error).

Objects may be duplicated in the Design Tree:

- Items included in macro elements are located in their places and in “*Macro*” section
- the surfaces which formed the body as a result of the “*Surface Knitting*” operation are located in section “*Surfaces*” and in the section related to the body.

The user can setup the Design Tree's composition by enabling and disabling sections' visibility; for assemblies, it is also possible to set up splitting components into the following groups: *Assembly units*, *Parts*, *Library components* and so on (see Section 2.1.2.3).



The Fig.2.1.2 shows display of creation sequence for the same part whose structure is provided in Fig. 2.1.3. In other words, these Figures demonstrate different representations of the same model in the Tree.

Model can include identical components. For example, creating a pattern enables creation of several identical components. Identical components form a branch in the *Components* section or in a group, if grouping is enabled. The branch name is formed according to a template. *<Component Name> (xN)* where N is the total number of identical components. Displaying of components grouped by types is illustrated in Fig. 2.1.4.

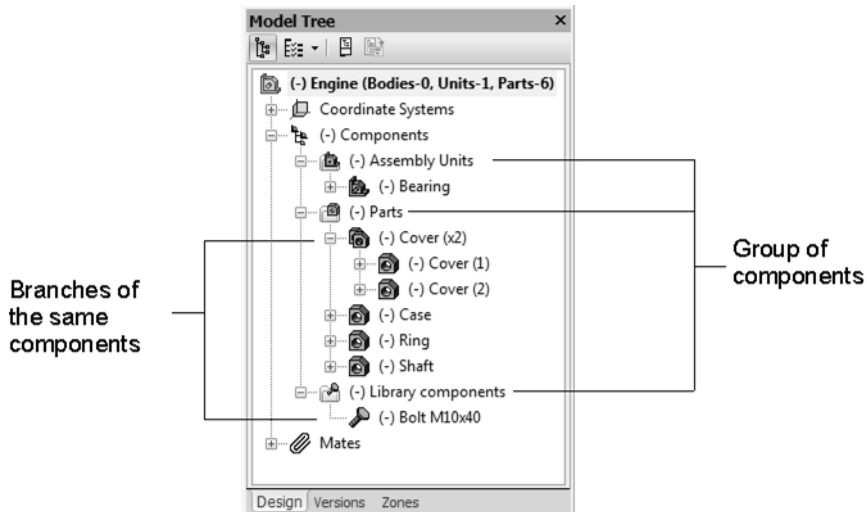


Fig. 2.1.4. Assembly structure with the display of groups

2.1.2.3. Setup of Design Tree display

You can enable and disable display of sections or component groups on the Design Tree, as well as select a default information presentation method, i.e. model structure or model creation sequence.



The **Model Tree Structure** button is located on the toolbar of the **Construction** tab, the **Model Tree** window; you can use this button to customize the default structure of the Design Tree and choose the Tree type to be saved.



This button is available if the **Display model structure** button is enabled in the toolbar of the **Construction** tab.

The setup dialog, which is described in Section 9.2.7.4.12, appears after the **Model Tree Structure** button is clicked.

Clicking the triangle on the right of the **Model Tree Structure** button expands the menu of this button which contains commands corresponding to sections' names (Fig. 2.1.5). To turn on/off the section display in the Design Tree, call the respective command.

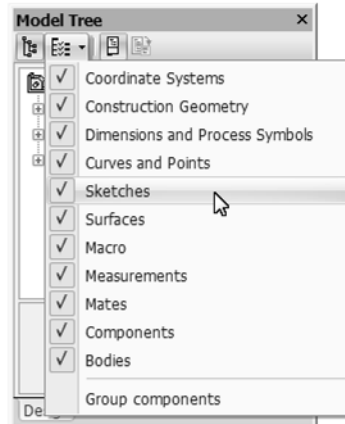


Fig. 2.1.5. Tree content setup

You can use the **Group components** *command* to divide the Design Tree's components into groups depending on type — Assembly units, Parts, Library components, and so on.

A tick on the left of a command's name means that displaying of the respective section or component grouping is enabled; if there is no tick, it is disabled.

Empty sections and groups cannot exist in the Tree. Therefore, the sections displaying of which is enabled or groups will appear in the Tree only after the first relevant item of this type has been created in the model. For instance, if a model does not have a single sketch, there will be no *Sketches* section in the Tree.

The objects in a disabled section remain in the model window (if their display is not disabled manually).

If a model is opened in several windows, the setting up of the Tree contents made in one of them is applied to all the windows the model is in.

2.1.2.4. Optional window of the Design Tree

Optional Design Tree window is a special window in which a part of the Tree is displayed (i.e. an object, a section, or relations). For instance, Fig.2.1.6 shows an optional window of the Design Tree which displays the "Sketches" section.

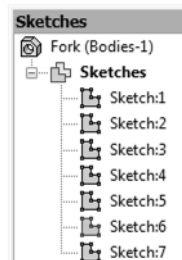


Fig. 2.1.6. Sample optional window of the Design Tree



In order to create an additional Design Tree window, in which an object or a section will be displayed, and click the **Additional Tree window** button on the control

panel of the Design Tree. Description of the Tree's additional window for an object or section can be found in this section.

In order to create an additional window of the Design Tree, select the required object in the Design Tree or in the model window and call the **Relations in Additional Window** command from its context menu. Viewing relations in the window is described in Section 2.1.2.7.2.

A copy of the selected part of the Design Tree is displayed in the additional window for the object or section. The header of the additional Tree window contains the name of the object/section shown in the additional window.

Changing the information presentation method in the Design Tree as well as Tree content does not influence the Tree optional window content. They can only be affected by changes in the model concerning the object/section that is being shown in the additional window.

For example, when the model structure was shown in the Tree, an additional window containing section "Sketches" was created. After this, the display of section "Sketches" in the Tree was disabled. The additional window still displays section "Sketches" with its content. Then, structure display was disabled in the Design Tree, which also disabled object grouping by section and displayed all the objects in the order of their creation. The additional window still displays section "Sketches" with its content.

Creating a new sketch in the model will cause a new object to appear in the additional window with section "Sketches". Deleting, hiding or excluding a sketch from the calculations will also be shown in this additional window.

If you delete the object (or all objects in the section) that is shown in the additional window, the window will be closed automatically.

The same object context menus are available in the additional window as in the Design Tree. Using these menus you can quickly call frequently used commands, e.g., Edit Object or Delete Object. Selecting an object in an additional window is equal to selecting it in the Design Tree.

The set of additional windows is remembered by the system before closing the model window, in which they were created. If you open this model again, as well as if you open it in a new window, additional windows of the Design Tree will not be available.

Additional windows of the Design Tree are useful while arranging quick access to frequently used model objects — sketches, construction geometry, etc.

For instance, by enabling model structure display in the Tree, you can create optional windows with "Sketches" and "Auxiliary geometry" sections. Later, you can disable displaying these sections in the Design Tree to shorten it or enable the model creation sequence in the Tree if it is more convenient.



If there are several additional Design Tree windows in the model window, the following location variants are recommended for them to use the screen area more rationally:

- Fix all additional windows at one of the borders of a document window and minimize them
- Fix all additional windows at one of the borders of a document window in such a way as to create a single bar.

2.1.2.5. Names and icons of items in the Tree

The name of each object can be entered during its creation. By default, it is assigned to objects automatically depending on their type (*Part*, *Assembly*, *Assembly unit*, *Body* and so on) or method of their creation (*Axis through edge*, *Rotation operation*, *Chamfer*, *Coaxiality (Cap – Gasket)* and so on).



Names of parts and subassemblies are taken from files of those components.

Many items of the same type can exist in a model. For recognition purposes, a sequential number of the given item type is added automatically to item names which were assigned by default. For instance, *Fillet:1* and *Fillet:2*, *Trim by plane:1* and *Trim by plane:2*.

You can change the objects' names:

- for the model's root object, components and bodies — using the command for assigning properties of a model, component, body (see Section 5.1.4.1.1).
- for other objects — during their editing (on the **Properties** tab of the Property bar).

You can also rename any object directly in the Design Tree. For this, do the following.

1. Select item name item in the Design Tree.
2. Click on the name selected or press <F2>.
3. Enter the new item name.
4. Click outside the tree items list or press <Enter>.

The new item name will be saved in the Design Tree.



For a model's root object and components, you can enable displaying names consisting of two parts, *name* and *designation* (see Section 9.2.7.4.13). The designation should be set as an object's property to be displayed (see Section 5.1.4.1.1). When renaming an object in the Tree, you can only change its name, not the designation of it, even if it is displayed.

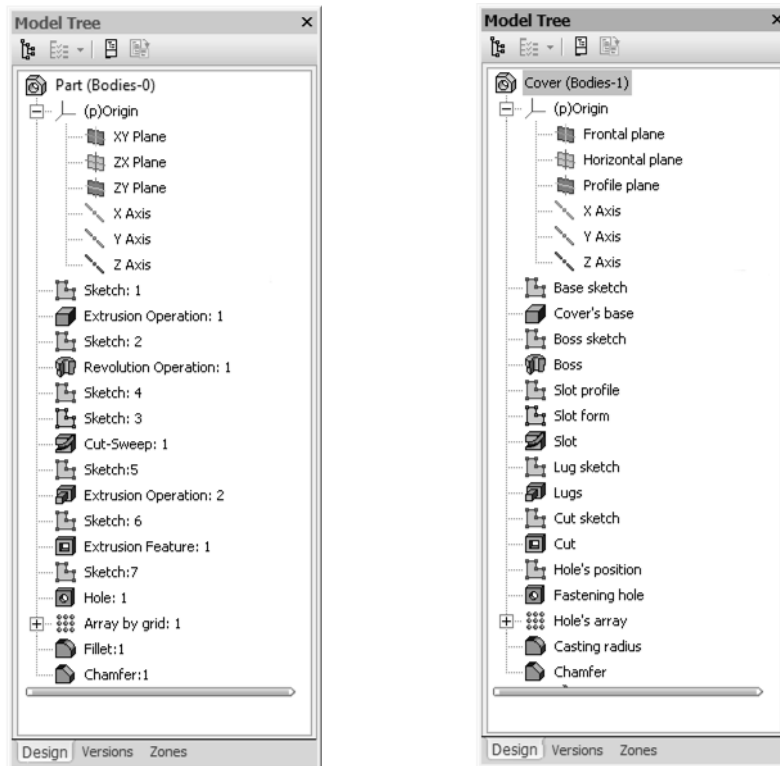


Fig. 2.1.7. Renaming objects in the Design Tree

An icon is displayed to the left of each item in the Tree. It corresponds to the item creation method. Unlike the object name, the icon cannot be changed. Due to this, visual information about creation method remains in the Design Tree after item renaming.

Usually, icons in the Design Tree are displayed in blue. If an object is selected, then its icon in the Tree is green. If an object was selected before executing the operation, then its icon in the Tree is red.

In the Design Tree, objects' icons may contain symbols indicating an object's state. For example, a read-only indication may be added to a component's icon, error symbol and so on. After the object's state changes, the symbol disappears or will be replaced by a different one.

Additional designations are listed in Table 2 of Appendix I.

2.1.2.6. Pointer of model creation finishing

The Model creation finishing pointer is a horizontal line that terminates the Design Tree at the bottom or splits it in two portions.

The pointer of model creation finishing is displayed in the Tree if model creation sequence display is enabled in the Tree (see section 2.1.2.1).

Position of the model creation finishing pointer may be changed. To relocate the pointer, move the cursor to it. As the cursor becomes a two-pointed arrow, press the left mouse button. While holding down the button, move the pointer up or down.

Items which appear in the Design Tree beneath the pointer, will be conditionally deleted. These items and their derivatives will not be displayed in the model window, but the information about them will not be removed from the document. The icons of the excluded elements are displayed in gray color and marked with a “lock” icon in the Design Tree.



Instead of moving the Pointer by the mouse, you may use the **Pointer under selected item** command. The command is called from the context menu of the item selected in the Design Tree.

Moving the pointer by the mouse allows to bring it next to the top border of Design Tree window; however, it can actually be placed only in between operation icons.

Upon every pointer relocation in the Design Tree, the model is updated.

To quickly move the pointer to the end of the Design Tree, you can use the **Roll-back Bar to End of the Tree** command from the pointer context menu.

After the pointer has been moved down, the conditionally deleted model items will be restored.

2.1.2.7. Hierarchy of model objects

Hierarchy of model objects— the order of their submission.

To create any model object, there should be used already existing objects (for instance, to create a sketch, a plane or face is required, to create a chamfer — an edge etc.).

An object, acquiring under its creation certain parts and/or properties of another object, is considered **dependent** to this other object.

For example, the sketch is built on a extrusion feature face — then, the sketch is dependent on the feature. A sketch contains projections of edges of the forming element — the sketch is dependent on this feature. The cut forming element is created by an operation with a sketch — the element is dependent on the sketch. When creating a forming element, its extrusion depth was defined up to a vertex of a rotative element — then the extruded feature is dependent on the revolved feature. A chamfer was created on an edge of a sweep — then the chamfer is dependent on the sweep. The construction axis is drawn through vertices of forming elements — the axis becomes dependent on those elements. The construction plane is drawn through the axis perpendicular to a element face — the plane becomes dependent on the axis and the forming element. This repeats on and on.

There are two types of relations between objects in the KOMPAS-3D model object hierarchy.

- If the object is subordinate to another object, it is called **derived** with respect to the subordinating object.

- If the object is subordinating to another object, then the subordinating object is called **original** with respect to the subordinate.



In certain systems of 3D-modeling the original objects are referred to as “parents”, and the derived objects — “children”.

Coordinate system, existing in the model from the very creation, always is original object (you can create the first sketch and other model objects only on base on it or its elements— coordinate planes) and never shall be the derived object (its parameters do not depend on other objects).

The last object in the Design Tree is never original (since no objects were created after it that could be based on it).

Any other objects can be either original or dependent. The same item can be a derivative and original at the same time but for different elements. For example, a hole is a derivative element of its own sketch and an original element for a chamfer created on that hole’s edge.

An element is always dependent on one or several elements that are above it in the Design Tree, and can be a parent to one or several elements, which are below it in the Design Tree.

However, this rule does not uniquely define relations between particular objects. Object position does not allow you to judge, which of them is original and/or derived with respect to this object.

If necessary, you can view relations of any objects in the model (see Section 2.1.2.7.2).

2.1.2.7.1. Hierarchical relations between the objects

Hierarchical relations between model elements are an inseparable part of this model. User can not refuse to make these relations or delete them.

Relations occur in the course of model object creation and will persist unless those objects are deleted or edited. Hierarchical relations are maintained through any model changes.

For example, when creating a sketch on a feature face, the hierarchical relation between sketch and face is created. As a result, this sketch will stay on its “home” face through any changes to the model (up until deleted or moved to another face).

Any item is involved in hierarchical relations together with its original and derivative elements. Relations have the following properties:

- At changing the parent object the derivative one changes too
- The derivative object can be changed both by editing of the source object and by changing own, independent parameters of this derivative object.

2.1.2.7.2. View of Objects Relations



To view relations involving a particular object, press the **Relations button** on the toolbox of the Design Tree. The relation view window shall appear at the bottom

part of the Tree. Then select the object needed in the Design Tree or any part of it (e.g. face of a shape-generating element) in the Model window. In the relation view area the information on the hierarchy of relations for the selected object will be shown (Fig. 2.1.8).

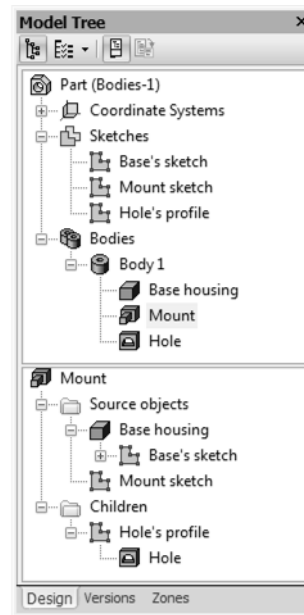


Fig. 2.1.8. View of hierarchy

The first line of the relations area shows the name of the element whose relations are considered.

The two sections subordinating to shown element display elements belonging to the described element's hierarchy as structured lists. Sections in these lists can be expanded and collapsed by clicking the mouse on the "+" and "-" signs next to their names. To view long lists, you can use scroll bars.

The **Source objects** list displays source objects, the **Children** list — resulting ones. The names of objects in the relation window are the same as their names in the Design Tree (if you entered new names of elements instead of the generated by default, these names will be shown in the relation window).

First level of **original** object list contains elements directly subordinating the current one. If these elements are in turn dependent on other elements, then the next layer of the list contains their original elements.

First level of original object list contains elements directly **subordinating** the current one. If these elements are in turn parents to other elements, then the next layer of the list contains subordinate derivative elements.

Therefore, the window allows tracking not only direct (explicit), but also indirect (implicit) dependency relations.



The hierarchy of object relations can be viewed in a particular window of the Design Tree (Fig. 2.1.9). To do this, you should select an object in the Design Tree or in the Model window and apply from the context menu the command **Relations in Additional Window**.

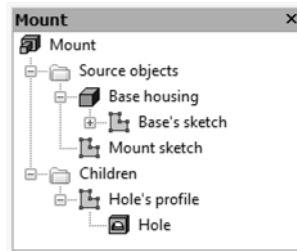


Fig. 2.1.9. Viewing of hierarchy in an additional window of the Design Tree

One would commonly need to know an element hierarchy in order to determine what elements changes (editing or deleting) could directly or indirectly affect the given element, and what elements can be affected by changes in the given element.

Let us see an example of definition of hierarchical relations of the object.

Fig. 2.1.8 and 2.1.9 show the element relations hierarchy *Boss*.

Boss, judging by its icon — an attached extrusion element.

What objects are original for *Boss*? To answer this question, let us analyze the structured list **Source Objects**.

At the level of the list, directly following the *Boss*, there are *Boss Sketch* and *Framework Base*. These objects are sources for the *Boss* because they are directly related to it: the sketch has been used for generating the extrusion element, and the framework base — for automatic definition of extrusion depth (to execute extrusion operation, option **To Vertex** has been enabled and the base vertex specified).

Framework Base, to judge by its icon, is an extrusion element. This extrusion element is based on the *Base Sketch* (it is located on the next hierarchic level).

Base Sketch, in its turn, was created on the *Plane XY* (it is located at the list's level, following the *Base Sketch*). *Base Sketch* and *Plane XY* are indirectly related to the *Boss*.

Boss Sketch was created on face *Framework Bases*. That is why the *Framework Base* is located at the level which follows the *Boss Sketch*.

The objects, being original for *Framework Bases*, has been considered above, their list is replicated following *Framework Base*. If the object is original for several other objects, it is (together with its original objects) replicated in the list (perhaps, at different levels) a certain number of times respectively.

Framework Base and its source objects are indirectly related to the *Boss*.

To see the derived objects of the *Boss*, let us analyze the structured list **Children**.

At the level, next to the *Boss*, an icon of the sketch, called *Hole Profile*, is located. This sketch is constructed on the *Boss* face and is indirectly derivative with respect to the *Boss*.

The *Hole Profile* has been used (judging by icon) for generating a cut rotative element — *Holes* (it is located at the list level next after the sketch). *Hole Profile* and *Hole* are indirectly related to the *Boss*.

There are no other objects in the hierarchy *Boss*.

Likewise, you can view hierarchical relations of any object of the three-dimensional model.



Hierarchy of relations is not fully displayed. This indicates that the sequence of model construction has been changed manually.

For example, the model had two extrusion elements, whereas the sketch of the second element was based on the face of the first one. Thus, the second element was derived with respect to the first one. The list of parent objects of the second elements included: its sketch, the first element, the sketch of the first element.

The first element is moved in the Tree to be lower than the second one (which is to be marked in the Tree as erroneous), then it is returned in place (then the error message disappears). As a result, only the sketch of this element remains as the parent object for the second extrusion element.

2.1.3. Model Display Control

2.1.3.1. Scaling, moving, rotating and rotation

You can control model scaling display as well as move, rotate and revolve the model.



If several model windows are open in KOMPAS-3D, each of them may have its own model position and image scale.

In some cases, results of changing model scale and position depend on the spatial location of its circumscribing rectangle. For instance, the model may revolve around the center of its circumscribing rectangle; Upon calling the **Show All** command, the scale of model display will change so that projection of its circumscribing rectangle on the screen surface will fill the model window.

Circumscribing model block is a conditional block whose faces are parallel to coordinate planes and pass through the points of model items which are most distant from each other.

You can select types of items which will be taken into account when creating a circumscribing block. To do this, call the **Service — Parameters... — System — Model editor — Dimensions command** and enable the required options in the appearing dialog.

2.1.3.1.1. Image scaling and moving

To change model scaling display and move, the following commands are used:



– **Move** command (see Section 1.4.1.3),

– Scaling commands (Section 1.4.1.2):



- **Show All**,



- **Scale Up in Frame**,



- **Zoom In/Out**,



- **Scale up**,



- **Scale down**,



- **Show sketch in full**,



- **Scale by selected objects**,



- **Previous scale**,



- **Next scale**.

The image moving command is in the **View** menu and the scaling commands are in the **View— Scale** menu.

The buttons to call the moving command and some scaling commands are on the **View** bar.

To quickly move the image (without calling the dedicated command), you can use **<Shift> + <arrows keys>**. Pressing any of these keys causes image to shift in the respective direction.

2.1.3.1.2. Revolve model

When modeling a part or an assembly, it is frequently required to view it from different sides. To rotate a model you can use:

- **Rotate** command,
- keyboard shortcuts,
- mouse.

Model rotation using the Rotate command



To rotate a model in the window, call the **View – Rotate**. This command allows rotating the model around the center of circumscribing block of the whole model or any of its objects, or around any object of the model. To do so, perform the steps described below after calling the command.

- **Rotate around the center of circumscribing block.**
Press the left button within the model window, the pointer will change—it will transform into a star with two arc-like arrows. Holding down the button, move the pointer.
 - **Rotate around a point** (vertex, center of a sphere).
Move the pointer to the desired object in the model window. As the object gets highlighted and the pointer takes the shape appropriate to the object being selected (see Table 2.1.8), click the left button. Then hold down the left button, and move the pointer (its shape is the same as when rotating around the center of the circumscribing block).
- The direction of rotation around the center point of the circumscribing block or around a point depends on the direction the pointer is moving (Table 2.1.3).

Tab. 2.1.3. Cursor movement dependence of model rotation direction

Cursor movement direction	Model rotation direction
Vertical	In vertical plane perpendicular to the screen surface.
Horizontal	In horizontal plane perpendicular to the screen surface.
By a diagonal	Direction is a resultant of relevant vertical and horizontal components.
Horizontal with pressed <Alt> key	Within the screen plane.

- **Rotating around a straight line** (axis, straight edge, conical or cylindrical face axis, tangent to a curve).
Move the pointer to the desired element in the model window. As the object gets highlighted and the pointer takes the shape appropriate to the object being selected, click the left button. An axis or curve tangent ghost will be displayed on the screen. Holding down the left button, drag the pointer (it has the shape of an axis with two arc-like arrows).
- **Rotate around a perpendicular to a surface** (face, coordinate or auxiliary plane).
Move the pointer to the desired point on the surface and click the left button. A ghost of the perpendicular to the surface in the given point will appear. Holding down the left button, drag the pointer (it has the shape of a plane with two arc-like arrows).
- **Rotate around the center of circumscribing block** (operation, curve, etc.)
Select an object on the Model Tree, then move the pointer into the model window. Holding down the left button, drag the pointer (it has the shape of two arc-like arrows).

To cancel selection of all the objects click **Re-select** on the Special Control Panel.



To exit the rotation command, click the **Interrupt command** or press `<Esc>`.

Rotating a model using the keyboard

To revolve the model around the center of a circumscribing block without calling a specific command, you may use the hot keys (the list is provided in table 2.1.4).

Tab. 2.1.4. Hot keys for model revolution

Hot keys	Direction of revolution
<code><Ctrl> + <Shift> + <^></code>	Up in vertical plane perpendicular to the screen plane
<code><Ctrl> + <Shift> + <v></code>	Down in vertical plane perpendicular to the screen plane
<code><Ctrl> + <Shift> + <right></code>	To the right in horizontal plane
<code><Ctrl> + <Shift> + <left></code>	To the left in horizontal plane
<code><Alt> + <right></code>	Counterclockwise in screen plane
<code><Alt> + <left></code>	Clockwise in screen plane
<code><Space> + <^></code>	90° up in vertical plane perpendicular to the screen plane
<code><Space> + <v></code>	90° down in vertical plane perpendicular to the screen plane
<code><Space> + <right></code>	90° to the right in horizontal plane
<code><Space> + <left></code>	90° to the left in horizontal plane
<code><Alt> + <^></code>	90° clockwise in screen plane
<code><Alt> + <v></code>	90° counterclockwise in screen plane

Model angle of rotation, upon enabling the `<Ctrl> + <Shift> + <arrow keys>` or `<Alt> + <arrow keys>` combination is called **model angle of rotation spacing**. Its value may be adjusted (see Section 9.1.11.5).

Rotating the model using the mouse

If you use a mouse with a scroll or a three-button mouse, you can rotate the model by dragging the pointer holding down the scroll or the middle button.

If the center of the circumscribing block is outside the borders of the screen (usually it happens when you have selected a very large image scale), the closest point of the model to the pointer is used as the center of rotation.

If this is not the case, rotation will be around the center of the block. To rotate around the closest point to the pointer, hold down `<Ctrl>`.



To make the model rotate in the screen plane, hold down <Alt> during rotation.

2.1.3.1.3. Model rotation

During parts and assemblies viewing, you can start continuous model rotation in the specified direction with specified velocity.



To start rotation, complete the following steps.

1. Call the **Rotate model** command .
2. Left-click the mouse in preview window, move cursor in the required direction of revolution and release the mouse button. Model will start to rotate around the center of the outline box in the specified direction.
 - 2.1. If you need to rotate the model around a point (a part's vertex, the center of a sphere, point), move the pointer to the desired element in the model window. The object will get highlighted and the pointer will take the shape appropriate to the object being selected (see Table 2.1.8). Click the left button, then hold it down and drag the pointer in the desired direction, then release the button.
 - 2.2. If you need to rotate the model around an axis, a straight edge, the axis of a conical or cylindrical face), move the cursor to the desired object in the Model window. The element will become highlighted and the cursor will take the shape that corresponds to the object being selected. Click the left mouse button. An axis or curve tangent ghost will be displayed on the screen. Click the left button, then hold it down and drag the pointer in the desired direction, then release the button.
 - 2.3. If you need to rotate the model around an axis passing through a given point of a face perpendicular to it, move the cursor to the required point in the model window and click. A ghost of an axis perpendicular to the surface running through it at the selected point. Click the left button, then hold it down and drag the pointer in the desired direction, then release the button.

Speed of the rotation will depend on how fast were the cursor movements when setting the rotation direction.

To cancel rotation, click the mouse in any part of the window. The model will stop. In this case, **Rotate Model** command is not interrupted and you can specify other direction and speed of rotation.



The **Rotate Model** command is the viewer application command. In order to use this command in KOMPAS-3D, add it to any menu or bar (see Section 9.4.2.1).

2.1.3.2. Model orientation

Position of a model in relation to the observer is called **model orientation**.



To change model orientation in KOMPAS-3D you, can use the model rotation command

Frequently, an orientation in which one of the projection planes is parallel to the screen plane is needed (in such a case, the model appearance corresponds to its image on a standard projection drawing, i.e., on the top or left view). It is hard to achieve such an orientation by rotating the model with the mouse. In this case, you can use a system-provided list of orientation names to change the orientation.

The **Orientation** button is located on the **View** toolbar. Clicking an arrow next to this button brings up a menu listing standard orientation names: **Top**, **Bottom**, **Left**, **Right**, **Front**, **Back**, **Isometric XYZ**, **Isometric YZX**, **Isometric ZXY**, or **Dimetric** (each of them corresponds to the respective viewing direction of the model).

Select a command corresponding to the desired orientation from this menu. 2.1.10). Model in window will be rotated in such a way that its position is in line with the specified sight direction.

The commands of this menu can be placed as buttons on a separate bar which then can be moved to any convenient location. To do this, drag the Orientation button by its title in any direction. The **Orientation** toolbar will be created (Fig. 2.1.11).

Note the difference of the **Orientation** bar from other toolbars: you cannot change the composition or the order of buttons on them.

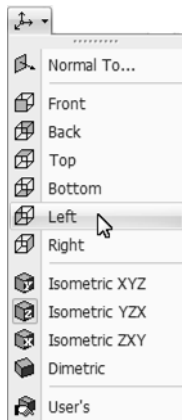


Fig. 2.1.10. Orientation name selection



Fig. 2.1.11. Panel **Orientation**

Sometimes, you may need to orient an auxiliary plane or a flat face of a part parallel to the screen rather than a projection plane. In order to set such an orientation, select the required flat item and choose the line **Normal to...** from the list of orientation names or from the context menu. The model will rotate in such a way as to make the view direction perpendicular to the selected object. In this case, normal vector of flat object will be pointed at the user.

Along with using standard orientation names, you may also save current orientation under your own name (see Section 2.1.3.2.1), and then come back to it anytime, by selecting this name from the list. You can save in a user-defined orientation information about the layers at the time of creating it. In this case applying the orientation will affect not only the position of the model, but the state of the layers as well.

2.1.3.2.1. Saving current orientation

To save the current orientation of the mode, proceed as follows.



1. Click **Orientation** button on the **View** panel.

The **View orientation** will appear on the screen, see Fig. 2.1.12.

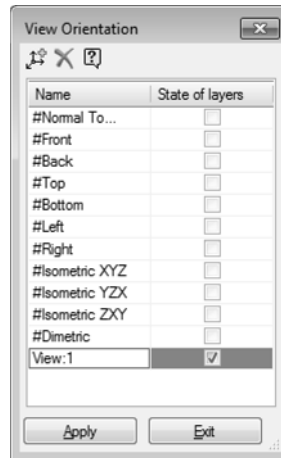


Fig. 2.1.12. View orientation selection dialog

The **Name** column contains a list of names of orientations existing in the model. The **State of layers** column shows whether the orientations store information about the layers. Layers in a model are discussed in detail in Section 2.15.13.



2. Click the **Add** button on the dialog toolbar. A new orientation will appear on the list. You can rename an orientation.

If a model, apart from the system layer, has at least one other layer, then by default the option in the **State of layers** column of the new orientation is selected. If the option is selected, along with information about the position of the model in the current orientation, layer information will be saved. To cancel saving layer status information, the option must be disabled.

3. Exit the dialogue box. To do so, click **Exit** button.

In the orientation selection dialog box, you may not only create a new orientation, but also select an existing one, as well as delete the orientation name created by user from the list.

- To select an existing orientation, focus on its name in the list and click **Apply**. The view will be re-arranged according to the sight direction specified.



- To delete an orientation from the list, highlight it and click the **Delete** button on the toolbar. The specified name will disappear from the list. You cannot delete standard orientation names (those starting with the “#” character).

User-defined orientation are also displayed on the orientation menu (Fig. 2.1.13). You may call the required command from this menu to setup a user-specific orientation.

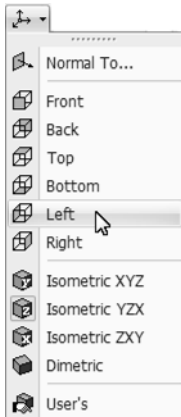


Fig. 2.1.13. New orientation in menu



If several model windows are open in KKOMPAS-3D, each of them may have its own model orientation.

2.1.3.3. Managing orientations in the model window

You can change model orientation by rotating using the mouse **orientation control element**, found in the left bottom corner of the model window. It consists of three thick arrows — red, green and blue, indicating positive directions of the X-axis, Y-axis, and Z-axis, respectively (see Section 2.8.3.1).

Initial position of the element corresponds to the **YZX Isometry** orientation. In all isometric orientations, the central point of an element is shown as a small cube, in others—a sphere (Fig. 2.1.14).

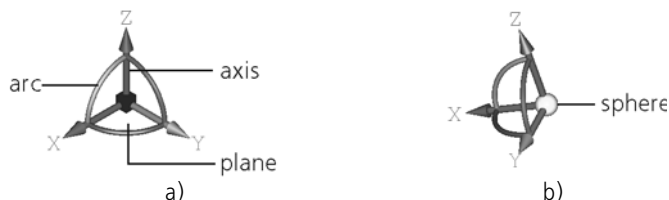
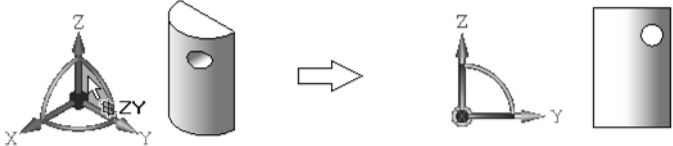
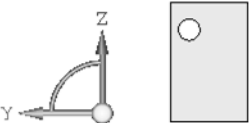
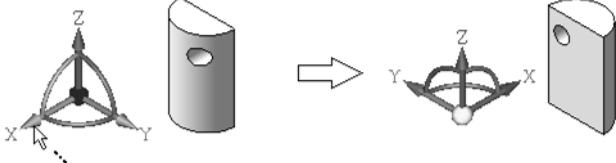
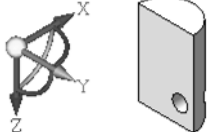


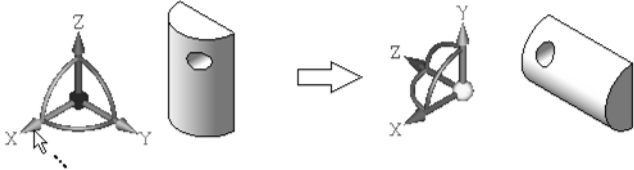
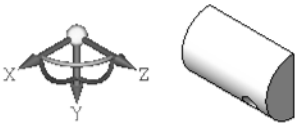
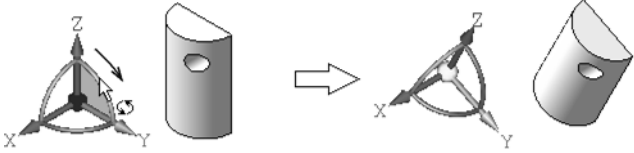
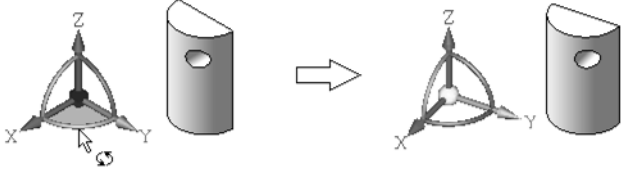


Fig. 2.1.14. Orientation control: a) in isometry, b) in any orientation

When the pointer is near the control, it activates and arcs appear on it. As you select arrows, planes or arcs on the element, they get highlighted, and the pointer changes shape. Clicking an arrow, plane, or arc (or rotating the arc by the mouse) rotates the model, clicking a sphere returns it to initial state. The techniques are detailed in Table 2.1.5.

Tab. 2.1.5. Orientation change techniques in the model window

Action	Result description
Click on arrow or perpendicular plane	Orientation front, back, top, bottom, left, right — the arrow points toward the observer . 
<Shift> + click on arrow or perpendicular plane	Orientation front, back, top, bottom, left, right — the arrow points in the opposite direction from the observer . 
<Ctrl>+ click on arrow	Rotate perpendicular plane 180° (around one of the axes of the plane): <ul style="list-style-type: none"> – if X arrow is selected—rotate plane YZ around the Z axis, – if Y—rotate plane ZX around the X axis, – if Z—rotate plane XY around the Y axis. 
<Ctrl> + <Shift> + click on arrow	Rotate the plane perpendicular to the arrow 180° (around another axis of the plane): <ul style="list-style-type: none"> – if X arrow is selected—rotate plane YZ around the Y axis, – if Y—rotate plane ZX around the Z axis, – if Z—rotate plane XY around the X axis. 

Tab. 2.1.5. Orientation change techniques in the model window

Action	Result description
<Alt> + click on arrow	Rotate the plane perpendicular to the arrow 90 ° around the axis. 
<Alt> + <Shift> + click on arrow	Rotate the plane perpendicular to the arrow 90 ° around the axis—in opposite direction. 
Arc rotation by the mouse	Rotate arc plane to an angle. 
<Ctrl>+ click on arc	Rotate arc plane with 15° step (by default). The step is set in the preferences (see Section 9.1.11.5). 
<Ctrl> + <Shift> + click on arc	Rotate the plane with a set step in the opposite direction. 
Click on sphere	Return to the YZX Isometry orientation position. 

Tab. 2.1.5. Orientation change techniques in the model window

Action	Result description
<Alt> + <Ctrl> + <Shift> + scrolling (when the pointer is near the element)	Enlarge/reduce the orientation control element.

During work with a model, e.g. when placing a component within an assembly, a similar control element will appear on the screen to change the position of the model. Working with the control is described in Section 2.11.2.5.3.


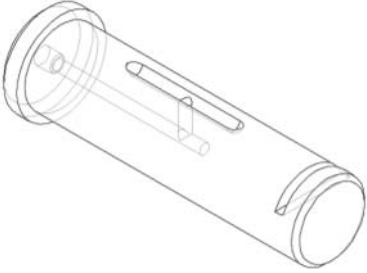



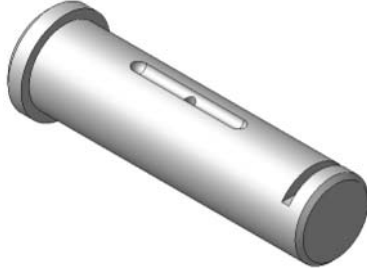
2.1.3.4. Model display

Several types model display are accessible in KOMPAS-3D. To setup the type of display, select its name in the **View – Display** menu, or press the relevant button on the **View** panel (see Table 2.1.6).

Tab. 2.1.6. Types of model display

Type	Description	Appearance
	Wireframe Range of all model edges and lines of model outline *.	
	Hidden Lines Removed Range of visible edges and visible parts of edges and lines of model outline.	

Tab. 2.1.6. Types of model display

Type	Description	Appearance
 Hidden Lines in Gray	Invisible edges, invisible parts of edges and invisible parts of outline lines are displayed in colors which differ from visible lines color * (the former are being lighter).	
 Display Shaded	Model surface display. Optical properties of surfaces (color, shine, diffusion etc.) are taken into account.	
 Halftone display With frame	Combining halftone display and display without invisible lines.	

* None of display types may influence model properties. For example, upon selection of the wireframe model display, the model remains solid filled with material (it does not turn into a set of "wire" edges), merely its surface and material are not displayed.



If several windows are opened in KOMPAS-3D, a different type of display may be enabled in each of them.

2.1.3.5. Perspective

Any optical instrument (for example, a human eye or a photo camera) captures the image of objects stretched in the instrument's axial direction with distortion, in other words, in perspective. In some cases, the perspective needs to be accounted for in order to obtain a realistic image of a three-dimensional model.



In KOMPAS-3D, the model perspective view display is supported. To get a model image that accounts for the perspective, call the **View** command. Relevant command button can be found on the **View** panel.

To switch off the perspective projection display of the model, release the **Perspective** button or repeatedly call the **Display – Perspective** command.

All types of displays listed in section 2.1.3.4 may be combined with perspective view



Fig. 2.1.15. Halftone perspective view of the model

2.1.3.5.1. Setting up options of perspective

The degree of the introduced image perspective distortion can be adjusted.

To do this, call the **Service – Parameters...**

The **Settings** dialog will appear on the screen

- If perspective view needs to be adjusted in the current window only, activate the **Current window** page and select **Prospective view settings** item,
- If perspective view needs to be setup in all newly opened windows, activate the **System** page and select **Model Editor— Prospective view setting** item.

The dialog has the single option— **Distance in Model Size**. It shows by how many times the distance from the part to the image plane is greater than the maximum part dimension. In other words, the model is shown on the screen in such a way that could be obtained by an optical instrument located at the specified distance from the model. The lesser the specified distance, the more noticeable is the image distortion.

2.1.3.6. Simplified model display mode

Parts and assemblies are displayed simplified by default.

Relatively small components and bodies of a part are replaced with rectangular blocks of appropriate size and colors in the following cases:

- Moving and rotating a model with mouse or by the commands **Shift** and **Rotate**,
- reorienting and rescaling (if the smooth transition is enabled, see Section 9.1.11.6)
- when setting small scale of display,
- moving and rotation of some components (under certain conditions).

Once the orientation or scale is changed or after zooming the display of model is restored. The mode of simplification works with all types of model displaying.



To disable simplified model displaying mode, call the command **View – Simplifications – Simplified displaying** or release button **Simplified displaying** on the **View** bar.

Settings for simplified displaying mode are changed in the dialog of system setup (see Sections 9.1.11.16.1 and 9.1.11.16.2).

2.1.3.6.1. Features of Simplifying Sub-assemblies

Displaying of sub-assemblies in the simplified mode has the following specifics.

- The rectangular box that replaces a sub-assembly is displayed in the color set for this sub-assembly when setting up its properties. Colors set for certain parts are not considered.
- If an assembly is exploded, the rectangular box that replaces the subassembly is located in the space according to the explosion parameters set for that subassembly. Explosion parameters set for certain parts in the sub-assembly are ignored.

2.1.4. General Working Techniques for Models

2.1.4.1. Selection of the objects

2.1.4.1.1. Object selection and indication

For execution of a number of 3D-element creation commands, it is necessary to **indicate** or **select** the objects on which this creation is based, such as sketches, vertices, edges and faces, auxiliary axes and planes, etc.

- Objects are **selected** when no three-dimensional construction command is active. Objects are selected in order to view them before calling a command. For example, an element needs to be selected in order to call the command of editing its parameters. In order to cancel selection of an object, you can click a vacant space of the model window and press <Esc>.
- Elements are **selected** upon defining parameters of the current command. For instance, after calling the lofted element creation command, specify the lofted sketches in succession.

To select or point an object, click it in the model window.

To execute the command, you often need to choose the object having certain geometric properties. For instance, to trim a model part by a plane, select the object that will serve as the trimming plane. Such object can be a flat face, a coordinate or an auxiliary plane.

Due to particular geometric properties, the object belongs to one of the types listed in Table 2.1.7.

Objects can be chosen in the model window (see Section 2.1.4.1.2) or in the Design Tree (see Section 2.1.4.1.5). The Design Tree makes it possible to choose the whole object and the model window makes it possible to choose separate ob-

ject parts, if any. For instance, to use the whole polyline as a path for a sweep operation, you should select it in the Design Tree, and to use one or more polyline segments, select them in the model window. One more example: by selecting an extrusion surface in the Design Tree, you can find all lines of intersection with its plane, and by selecting an individual face of this surface, you can find the line of intersection of the face with the plane.



Such object as a body can be selected in the Design Tree only. For this, the Tree must show model structure (see Section 2.1.2.2). The body can be selected not only in the Design Tree but also in the model window. Select a body face, edge or vertex for this and then call the **Select Body** command from the contextual menu.

Tab. 2.1.7. Types of model objects

Object type	Objects belonging to this type
Point objects	<ul style="list-style-type: none"> – Origin – geometric object handle in a sketch, – standalone point in a sketch, – standalone dimensional point, – 3D line vertex, – edge vertex, – group points, – points within point pattern instances, – control and connection points.
Straight Objects	<ul style="list-style-type: none"> – segment, – line in a sketch, – polyline segment, – coordinate or auxiliary axis, – straight-line edge.
Flat objects	<ul style="list-style-type: none"> – coordinate or auxiliary plane, – planar face, – sketch plane (for some commands).

2.1.4.1.2. Selecting objects in the window



















When cursor moves above the model, the system automatically carries out **dynamic targeting** of items.

The dynamic targeting is a search of the object under the cursor. As soon as the object is found, it is highlighted and its symbol (see Table 2.1.8) will appear near the cursor. As the cursor moves away, the highlight goes off, the symbol disappears, and dynamic targeting resumes.








The dynamic targeting during an operation is performed taking account of the objects' types. When the cursor hovers the model, the object of the currently required type will be highlighted.

To pinpoint or select an object in the model window, move the pointer to it. When symbol of the current object appears next to the cursor and the object itself is highlighted, click the left mouse button.

Tab. 2.1.8. Cursor appearance when selecting different types of items

	Object
	Vertex
	Edge
	Face or surface
	Axis
	Plane
	Component
	Body
	3D curve or edge
	Surface
	Sketch
	Thread Notation
	The origin of the absolute or local coordinate system
	Point Control or Attachment Point
	Linear Dimension
	Linear Dimension from Line to Point
	Radial Dimension
	Diameter Dimension
	Angular Dimension

Tab. 2.1.8. Cursor appearance when selecting different types of items

	Object
	Surface finish designation
	Datum Symbol
	Leader
	Marking symbol
	Stamping symbol
	Shape and position tolerance symbol.
	Ref.No. Leader

When selecting object while holding down the <Shift> key, you select a component in the model window whose element that selected object is or to which it belongs. In this way, you can select, for example, a whole part by picking one of its elements, such as a face, an edge or a vertex. If, while holding down the <Shift> key, you select any auxiliary object, then the parts or the subassembly will be highlighted in the model window to which that auxiliary element belongs.

Sometimes, to perform a command, you need to select a group of objects.

To select multiple objects in the model window (faces, stitches, auxiliary elements, etc.), hold the <Ctrl> key down while selecting them.

To make multiple parts current in the Model window, select them while holding down the <Shift> key.



Selection of item groups and part groups may be combined This means, for example, that you may select several objects by holding down the <Ctrl> key, and then, releasing this key, press the <Shift> key (the objects will remain highlighted) and select several parts while holding it. Thus, a group of objects and a group of parts will be simultaneously selected in the model window.

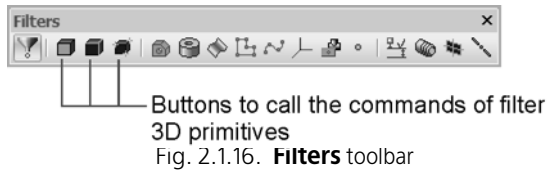
2.1.4.13. Object Filters

Sometimes, when the cursor is hovered over a model, the cursor traps several objects at once (i.e. a face and its edge), and the wrong item is highlighted.

Object filters are used to help selecting objects of the desired type. To enable them, activate the **Filters** panel (see fig. 2.1.16).



By default **Filter all** button is pressed on the panel. Enabling this button indicates that all vertices, edges, faces, axes, and planes, etc.— all objects shown on the **Filters** panel— are highlighted and may be shown (selected).



If, to perform your intended action, you need to specify (select) certain objects (or object primitives), push the respective button on the **Filters** bar. Once any of those buttons is pushed, the **Filter All** button becomes released.

If all buttons corresponding to object or primitive types are switched off, then the **Filter All** button activates automatically (in other words, you cannot fully disable the selection of all types of primitives and objects).

Filtering commands are conditionally divided into three groups:

- 1st group — filtering out 3D primitives: faces, edges, vertices,
- 2nd group — filtering objects that have primitives; these include components, bodies, surfaces, sketches, curves, points,
- 3rd group — filtering objects that do not have primitives; these include constructive axes and planes, layout elements.

Buttons for calling commands of this group are separated in the **Filters** bar.

You can select several objects for filtering by pressing respective buttons on the **Filters** toolbar. You can switch the filter buttons at any time while working with the model.

Filtration by several filters at once can be done using the following methods.

- **Filter combination.** If filters are selected from 1st, 2nd or 3rd group only, in the result you will be able to filter either all primitives, or all objects whose buttons are enabled. In this case, objects are selected wholly.

- Example 1. When **Filter Components** and **Filter Surfaces** buttons are pressed, all model components and surfaces are available for selection.

If an object is selected, the cursor looks as if during the dynamic search of a component or surface.

- Example 2. When **Filter Vertices**, **Filter Edges** buttons are pressed, all model vertices and edges are available for selection.

If an object is selected, the cursor looks as if during the dynamic search of vertices or edges.

- **Overlapping of filters.** Filters are selected from the 1st and the 2nd group simultaneously, in the result you will be able to filter the primitives of the selected type of objects. Not the whole objects will be selected but only their primitives — edges, faces, or vertices.

- Example 3. When **Filter Edges** and **Filter Components** buttons are pressed, only component edges are available for selection.

If an object is selected, the cursor looks as if during the dynamic search of edges.



- Example 4. When **Filter Faces** and **Filter Points** buttons are pressed, filtering is not performing, because point do not have “face” primitive.

Filters of the 3rd group do not take part in overlapping but ensure filtering as if in a combination of filters.

2.1.4.1.4. Selecting hidden, overlapping or closely located objects

Sometimes the object to be selected in the model's window is located close to other objects or overlaps them, or is hidden beneath them. At the same time, it is difficult (or sometimes even impossible) to select it with the cursor.

To select any of the nearby-located objects (including overlapping ones), use the object picking over mode. Picking over is possible when the System is waiting for instruction or selection of an item, and the cursor traps more than one item.

In order to select one of the hidden, coincident or closely located items, do the following.

1. Point the cursor to the group of objects containing the desired object.
2. Without selecting any of them, call the context menu command **Object Picking Over**. You can also press the **<Ctrl>+<t>** keys.
3. Pick over objects by pressing the **<Space>** key or calling the **Next Object** command from the context menu. Objects pointed at by the cursor at the time of calling the picking over command will be highlighted one after another.
4. When the desired object highlights, finish the pick-over mode by confirming the selection. To do this, call the **Select highlighted object** command from the context menu or press the **<Enter>** key. You can also click the mouse on the highlighted object or at any free space of the document window.
5. To exit the picking over mode without selecting any object, call the context menu command **Cancel Picking Over**. You can also press the **<Esc>** key.

If picking over was used to select an object while performing some command, the system will return to that command.

2.1.4.1.5. Selection in the Design Tree

Some items should be selected and specified not only in the model editing window, but also in the Design Tree.

To specify or select an object in the Tree, click the mouse on its name or icon.

In this way you can select or specify a sketch, plane, axis, shape-forming or constructive element (i.e., boss-revolved element, or hole, or fillet), component or mating.



Picking and specifying objects in the Tree can be done only in the three-dimensional construction mode. If the system is in the sketch mode, then picking and specifying objects in the Design Tree is impossible (even though the Tree is displayed on the screen).

When specifying or selecting any item in the Tree, the relevant part of the model is highlighted or selected in the window.

If mate is selected in the Tree, the items are selected in the model window involved in the mating.

In order to select several items in the Design Tree, specify them by holding the <Ctrl> key.

To select multiple objects following one after another in the Design Tree, select the first (last) of them, press and hold the <Shift> key, and then select the last (first) object. The selection will be expanded to all objects in the group.

After the item has been selected using any method (including in the model window), the relevant blue icon in the Design Tree will turn green. For example, when selecting an edge, the icon of the operation that produced the edge will change color, and when selecting a plane, the icon of this plane will change color.

Once an object was selected by any available means, its respective icon in the Model Tree turns from blue to red. For example, when selecting a face, the icon of the operation that produced the face will change color, and when selecting a sketch, the icon of the sketch will change color.

Components are shown on the Design Tree as icons. There is a “+” sign next to an icon on the left side. This means that the list of items comprising the component is minimized. Therefore, the items which comprise the component may not be visible in the Tree even if they are selected in the model window.

Use **Service — Show in Tree** command to see the item in the Tree which is selected in the window model. After calling the command, the icon of the respective feature is highlighted green, while the Design Tree expands to bring the item into the view.



If the item belonging to a shape-forming element (for instance, face of extrusion element), is selected in the model window, upon calling the **Show in Tree** command the icon of relevant shape-forming element will be selected in the Design Tree, once the command Show in Tree is called.

2.1.4.2. Managing operation's parameters

2.1.4.2.1. General Information

In KOMPAS-3D, model's objects are created and edited by performing operations. Most operations have a certain set of numeric parameters. For example, numeric parameters of the extrusion operation are extrusion distance, inclination angle, and wall thickness; the numeric parameter of the scaling operation is the scaling factor.

A 3D object **handle** corresponds to each numeric parameter of operation. Handles are displayed as black squares when an operation is performed.

If a value of the operation's parameter is expressed in linear or angular values, the parameter is displayed as an (**operation's dimension** of the respective type).

In the examples above extrusion distance, inclination angle, and wall thickness are given as an appropriate dimension and a handle; scaling quotient is shown as a handle only.

Some operations are created on the basis of a sketch. If the user assigns dimensions to a sketch, these dimensions will be displayed during creation of the operation.



Dimensions of sketches and operations are displayed during creation or editing of operations only if the **Dimensions of sketches and operations** option is enabled in the dialog for setup of dimensions and designations (for more information about the setup of displaying dimensions and designations, see Section 9.1.11.12).

For more information about dimensions of sketches and operations, refer to section 2.9.2.2.

In Fig. 2.1.17, you can see an example of the extrusion element creation based on sketch. The following is displayed in the element's ghost:

- handles – as black squares,
- operation's dimensions – extrusion distance (25), inclination angle (10°), wall thickness (12),
- sketch dimensions – height (60), width (76).

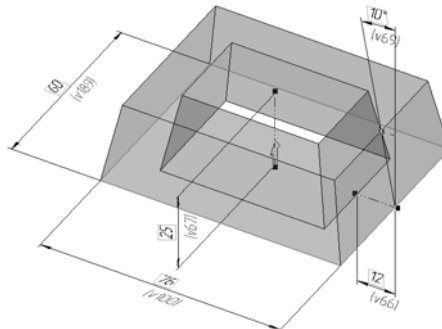


Fig. 2.1.17. Example of the extrusion element creation based on sketch.



For each numeric parameter of the operation, a **variable** is created automatically. A sketch dimension variable can be created by the user (see Section 7.2.2.20). variable names are displayed beneath the dimension lines. If variable names are not displayed, call the command **View – Names of variables in dimensions**.

A number or a constant can be the value of the numeric parameter. Besides that, the parameter's value can be set by a mathematic expression.

In order to set numeric parameters of an operation, you can use the following methods:

- entry of the parameter's or expression's value for calculation of the parameter's value in the respective field of the Property bar (see Section 1.4.2.5 for more in-

formation about entry of values; information about the syntax of expressions can be found in Section 7.1.3.3.1);

- moving a handle with the mouse (see Section 2.1.4.2.2);
- setting the value of the sketch or operation's dimension (see Section 2.1.4.2.3).

If the numeric parameter of the operation is displayed as a dimension, you can assign a **tolerance** to the value of this parameter. Assigning a tolerance to the operation's parameter value is described in Section 2.10.2.2.

Numeric values of operations' parameters can be modified after creation of the operation. For more information about editing of operations' numeric parameters, see Section 2.14.3.3.

2.1.4.2.2. Objects' handles

During construction of objects, use of handles allows to change their various parameters (sizes, position, shape etc.) without keyboard, but by moving the points with mouse.

Handles of three-dimensional objects correspond to numerical parameters of the operation (numeric fields located on the Property bar). Handles are displayed as black squares.

Certain objects (for instance, spirals, extrusion features, etc.) have an abundance of numeric parameters. All handles of such objects cannot be displayed at the same time, and therefore, only the points corresponding to the numerical fields in the current Properties Toolbar tab are shown on the screen. As an example of such an object, let us consider a conical spiral. Right after calling the **Conical Spiral** command, the **Construct** tab activates on the Properties Toolbar. It has four numerical fields, and, therefore, the spiral ghost has four handles (Fig.2.1.18, a). When switching to the **Diameter** tab, these handles disappear, and the other ones corresponding to the numerical fields of the **Diameter** tab (Fig.2.1.18, b) come up.

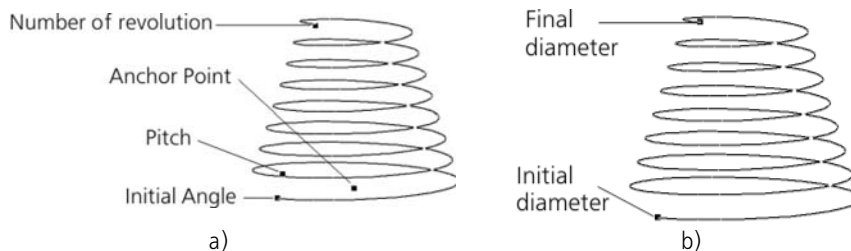


Fig. 2.1.18. Handles of a conical spiral:

a) the **Construct** tab on Properties toolbar is active; b) the **Diameter** tab on Properties toolbar is active

To modify a parameter value, you need to activate its respective handle and move it.


To activate the point, move the mouse cursor to it. Once the point is selected and a label with the parameter name and value of the parameter appears next to it (or an expression for calculation of the parameter's value), click the left mouse button.

Drag the mouse while holding the button down. The selected handle will follow the cursor, and the respective parameter value will be changed and displayed in the text near the cursor. The object ghost will be updated dynamically. Once the desired value is reached, release the mouse button.

Some parameter values can be counted in either direction from a neutral position. When dragging a handle corresponding to such a parameter, the appropriate direction switch will be automatically activated on the Properties Toolbar. For example, an extrusion side wall draft can be directed inward or outward. When dragging an *Angle* handle from the zero position inward to the body, the **Draft Inside** switch will be activated, and when dragging it outward, the **Draft Outside** switch will be enabled.

It is more convenient to control a changing parameter value while dragging a handle in the Rounding (value approximation) mode.



To turn on this mode, press the  button on the **Current Status** Panel. When in the Rounding mode, the parameter corresponding to the handle to be moved may take only such discrete values that are multiples of the current cursor step. The value of the current cursor step is displayed in the same-name field on the **Current Status** panel. In this field, you can change— by entering from keyboard or by selecting from the drop-down menu— the cursor current step for the active window. For a shortcut to activate this field, use the **<Shift> + </>** combination (**</>** key should be pressed on auxiliary numeric keypad). The list of steps and default step can be specified in the cursor setup dialog (see Section 9.1.7.1).

2.1.4.2.3. Setting the value of operation's dimensions

One of the methods to setting the value of an operation's numeric parameter is setting the value of the dimension of operation.

In order to set the value of operation's dimension, do the following:

1. Call the dialog for setting the dimension's value. For this purpose, double-click the dimension text of the dimension in the model window.
2. Enter the dimension's value in the **Expression** field of the appearing dialog: a number, constant, or expression for calculation of the dimension's value. Syntax of expressions is described in detail in Section 7.1.3.3.1.
3. Click **Create Object** button. The dialog will be closed, and the model will be rebuilt according to the value of the parameter.

More information about the dimension value setting dialog and its usage can be found in Section 2.9.2.2.1.

The value of the sketch's dimension is set in the same way as the value of the operation's dimension.



Setting the value of the sketch's dimension is only possible if the dimension is the associative one.

2.1.4.2.4. Peculiarities of setting numeric parameters of operations in the Property Bar fields

When entering values of the operation's numeric parameters in the Property bar fields, the following peculiarities should be taken in to account.

- If a formula is input in a field, as you type it the field will display the formula, and after confirming the input (by pressing *<Enter>*)—the result of calculating it.
- The field of a parameter given as a formula has a yellow background color. To get information about the parameter, move the pointer to it. A box with the name of the parameter, name of its variable, the formula to calculate the parameter and the calculated value, will appear next to the pointer. If a new operation is created (not an existing one being edited), the text box will have no variable, since the variables corresponding to operation parameters exist in the model only after the completion of the operation.
- If a value given as a formula can be changed using a counter or a handle, a free term is added to the formula. The value of the constant term is equal to the value of the change. The sign in front of the constant term depends on whether the parameter's value increases or decreases.
- If the formula for calculating a value contains a variable non-existent in the model, after accepting the input the variable will take its default value, and the value of the operation parameter will be equal to the result of calculating the formula with the new variable. After creation of the operation, the new variable will automatically appear in the model, while its name and default value will be displayed in the main section of the Variables Window (for more details about variables, see Section 7.1).
- If only the name of the variable non-existent in the model is input in the field, the variable will take the previous value in that field. If the field contained an expression, the value of the new variable will be equal to the result of the expression's calculation. After creation of the operation, the new variable will automatically appear in the model, while its name and value will be displayed in the main section of the Variables Window.

2.2. Sketches

2.2.1. Work with a sketch

2.2.1.1. General Information on Sketches

Sketch is a 3D-modeling object, created by using tools of a drawing graphical editor. A sketch may be positioned on a coordinate or auxiliary plane, as well as on a flat face.

Sketches are used for different purposes, for example:

- ▼ setting the section form of a body or surface,
- ▼ path of the section movement construction
- ▼ Pattern instances position construction.

It is possible to use the same sketch in multiple different operations.

A special mode of working with model is used for working with sketches – **sketch mode**.

2.2.1.2. Sketch mode

Sketch mode is a special KOMPAS-3D three-dimensional model operation mode. You can switch to this mode when creating a new or editing an existing sketch.

Working in the sketch mode is practically the same as working with a KOMPAS fragment. The difference is that it is impossible to create such objects as technological designations and tables, hatches, fillings, break lines and some others. This is due to the fact that, unlike graphic objects, they are not involved in creating the three-dimensional feature geometry when sweeping the sketch and, unlike dimensions, they do not define the sketch configuration.

By default, the **parametric mode** is enabled in a new sketch. For detailed aspects of working in this mode, see Section 7.2.1.7; for detailed information about parametric mode setup in a sketch see Section 9.2.7.7.1 on p. 1864.

Commands for 3D modeling are not available in the sketch mode. Model objects can be used in a sketch for projecting or snapping (see Section 2.2.2.3 on p. 146).

When you finished working with a sketch, exit the sketch mode to continue creation of a model. See Section 2.2.1.4 for more information about using sketches in operations.

2.2.1.3. Sketch status diagnostics

If the sketch is a parametric one, it can be in one of the three states: **fully defined**, **not fully defined** or **redefined**. These statuses are indicated by symbols in front of sketches' names in the Design Tree:

- (+) — fully defined sketch, i.e. its objects having no freedom degree in the sketch coordinate system,
- (-) — not fully defined sketch.
- (!) — redefined sketch.



It is recommended, that all sketches in a model were fully defined. In this case, the equation systems of sketches are solved in a more stable fashion.



To view or edit sketch state, enter into its edit mode. Enable display of object constraints (see Section 7.2.5.1 on p. 1620) and degrees of freedom (see Section 7.2.5.2 on p. 1624), if it is disabled.

- ▼ If a sketch is fully defined, none of its objects has degree of freedom, i.e. if degrees of freedom are enabled, their symbols are not displayed.
- ▼ If a sketch is not fully defined, then at least one of its objects has at least one degree of freedom. In order to fully define a sketch, relation and/or limitation objects should be applied to it in such a way, so that none of the objects have any degree of freedom.
- ▼ A sketch is considered to be redefined, if surplus relations and/or limitations are applied to its objects (while objects' degrees of freedom may be present or absent). Surplus relation and/or limitation symbols are displayed in brown color. For example, a surplus relation is the *Parallelism* applied to segments that already have the *horizontality* limitation.

If a sketch is non-parametric (i.e. its objects are neither not related to each other nor to model objects and have no limitations), no symbols are used in front of its name in the Design Tree.

2.2.1.4. Using a sketch in operations

Outside the sketch mode, only some of graphical objects built in it are displayed – those that lie in the current layer and in active layers (there are may be several layers in a sketch, as well as in a fragment, see Section 3.5.6 on p. 1181) and are of one of the following types and/or styles:

- ▼ point,
- ▼ Centerline,
- ▼ Line with the *Basic* style,
- ▼ Line segment with *Axis* style.

When performing operations that use a sketch, only those its graphical objects are taken into account that are visible outside the sketch mode.

Dimensions that are set in the sketch are available when performing an operation and when enabling special mode (see Section 2.9.2.2 on p. 632).



Thus, if auxiliary objects are required for creating an image in a sketch (especially, parametric) that should not be taken into account when performing an operation, use the line style for them that is other than *Basic* or *Centerline*.

Move the dimensions, that should not be visible outside the sketch mode, to a hidden layer.

A sketch specified for using in an operation is interpreted by the system as one or several **contours** made of the sketch's lines.

Note that sketch's lines are always fully included in the sketch. For example, if two segments are created in sketch, and they connect with each other at the vertex, they make up the same contour, while if segments intersect with each other, there are will be two contours. Also, two contours are made by three segments with a common vertex.

If sketch's lines form several contours, correct performance of the operation usually requires that the contours have no common points, do not intersect or be tangent to each other. There are more requirements to sketches. These requirements depend on operations that use sketches and are described in sections dedicated to operations.



In some operations, you can use both the whole sketch (i.e. all of its contours), and in this case the sketch has to be specified in the Design Tree, or only some lines of the sketch, in which case they have to be specified in the Model Tree.

2.2.1.5. Selecting a plane for sketch construction

Operation sketch plane selection does not affect further steps of working with the model and its properties.

However, when selecting a plane for a sketch of the first body or surface, it should be borne in mind that object's position in the standard orientation will depend on this selection. For example, if section sketch of an extrusion body is constructed in the XY plane, then the body projection on the front view will coincide with the shape of the sketch (Fig. 2.2.1).

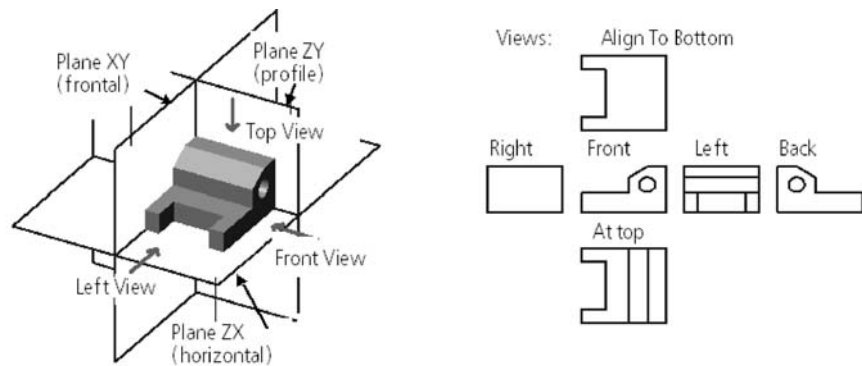


Fig. 2.2.1. Position of part relative to coordinate planes

2.2.1.6. Sketch from Library

Instead of creating a new image in the sketch from scratch, you may insert objects already available in the sketch library into it (for details, see Section 12.1.4.2 on p. 2044).

2.2.1.7. Orientation of sketch's plane and sketch's view scale

It is convenient to construct a sketch if its plane coincides with the plane of screen. If the sketch plane is perpendicular to the screen plane, construction is impossible.

By default, the system is set in such a way that when entering the sketch mode the model is automatically turned into **Normal to...** orientation relative to the sketch plane, and, upon exiting the sketch editing mode, it shall return into initial position.

The sketch's view scale is specified in the following way:

- ▼ when a new sketch is created, it is equal to 1 (if the sketch is created in the coordinate plane) or selected in accordance with the object's dimensions (if the sketch is created on an auxiliary plane or flat face),
- ▼ if an existing sketch is edited, it is selected by the image's dimensions in the sketch, i.e. the **Show the sketch in full** command is performed automatically.

Automatic setting of orientation and scale can be modified in the display settings dialog (see Section 9.1.11.5 on p. 1728).

If the model automatic reorientation is disabled, then the sketch plane can be manually reoriented parallel to the screen.

To do this, command **Normal to...** is used.

You can set the sketch plane into **Normal to...** orientation both before and after switching over into sketch mode. In the first case, you should select the plane of the future sketch and call the **Normal to...** command from the context menu. In the second case, selection of the plane is not obligatory—you may simply apply the **Normal to...** command from the context menu (it is accessible, if none of the geometric construction commands is active) or from the **Orientation** button menu.

2.2.2. Sketch creation

2.2.2.1. Sketch creation process

The sketch is positioned on a flat object — coordinate or auxiliary plane or flat surface. This object can be specified both before entering the sketch mode and afterwards.



1. Call the **Operations – Sketch** command or click the **Sketch** button on the **Current state** bar. The **Sketch** button will remain enabled. It means that the system is in the sketch edit mode.
 - ▼ If a flat object is selected prior to calling the command, the sketch will be created within this plane and the sketch's coordinate system will have the default position.
 - ▼ If no flat object have been selected, the process of sketch positioning will be run automatically (see Section 2.2.2.2 on p. 145). Specify the required flat object.

The operation's icon appears in the Design Tree. It is marked with the “lock” icon denoting that the sketch is currently being edited.

The compact bar and Main Menu will be modified. Commands for creation and editing of graphic objects, selection, measurements, dimension settings, parametric relations and limitations will become available.

The sketch's coordinate system will be displayed on the screen.

2. Create the required image in the sketch.



You can import a draft into a sketch from a previously created drawing or fragment via the clipboard. This helps to rely on existing drafting design documentation when creating a three-dimensional model.

- To return to the mouse of 3D construction, call the **Operations — Sketch** command again or release the **Sketch** button in the **Current status** bar or click on the sketch mode icon in the document window.

Right after leaving the sketch mode, the new sketch is selected in the graphic area.



Commands for running operations that can be performed based on the created sketch can be called directly in the sketch mode. For example, having created an extrusion body section in the sketch, you can immediately call the **Extrusion operation** command. The sketch edit mode will be finished automatically, and the extrusion body creation command will be run.



A sketch is an object that uses coordinate system (for object coordinate system, refer to Section 2.8.3.4.1 on p. 598).

2.2.2.2. Sketch Positioning

Sketch positioning is the selection of a plane for it and setting the position of its coordinate system on this plane.

The sketch positioning process starts after calling the **Sketch** command if no plane or flat face had been specified before running it.

By default, automatic object creation is enabled during the sketch positioning process (see Section 1.4.2.10 on p. 93). Therefore, once the plane or flat face for sketch positioning is specified, the system will immediately enter the sketch construction mode. If you need to perform other operations in addition to specifying a flat object, such as setting the position of the sketch's coordinate system or disabling the sketch's link to the plane in which it is positioned, switch off automatic creation by disabling the respective button in the Advanced Control Panel.

In the process of sketch positioning, a ghost of the sketch's coordinate system is displayed in the graphic area while the Property bar looks as is shown in Fig. 2.2.2.

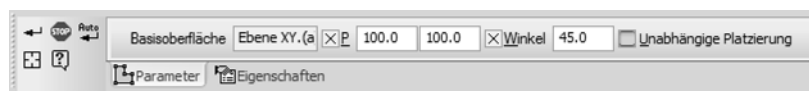


Fig. 2.2.2. Property bar in the process of sketch positioning

In order to define a plane for sketch positioning, specify an auxiliary or coordinate plane or a flat face. The selected object will be highlighted in the model window, while its name will appear in the **Base plane** field in the Properties ar.

In order to move or turn the sketch's coordinate system with respect to its initial position, enter values for coordinates and angle in the **Snap point** and **Angle** fields in the Property bar. Change of coordinates and angle will be reflected in the sketch's coordinate system ghost, in the graphic area.

The coordinate system of the sketch may also be moved as needed or turned by the mouse. For this purpose, cancel fixation of the **Snap point** or **Angle** field in the Property bar and then specify the new position or rotation angle of the sketch's coordinate system in the graphic area.

If you do not need the sketch to be related to the base plane, enable the **Independent Location** option. The sketch will keep its position in space, but its link with the base plane will be broken (the base plane will no longer be highlighted in the Model Window, and its name will be deleted from the **Base Plane** field).



Click **Create Object** on the Advanced Control Panel in order to confirm the position of the sketch.

2.2.2.3. Using model objects when creating a sketch

When working with sketch, you have the following options.

- ▼ **Snap to model objects** when creating geometric objects and dimensions. In this case, auxiliary projection of the model's object is formed in the sketch. The created object is parametrically linked to the projection. Snapping to model objects is described in Section 2.2.2.3.1 on p. 146 and construction projection — in Section 2.2.2.3.2 on p. 147.
- ▼ **Use model objects as base items** when creating geometric objects and dimensions. To do this, just indicate model objects when creating/editing the image in the sketch. As a result, a construction projection of the model object is generated and the sketch object is linked to it. Examples: a segment parallel to the edge parallel; the angular dimension between the projection of a straight object¹ and the segment, the *equality of lengths* link between the segment and the projection of a straight object. Construction projection is covered in Section 2.2.2.3.2 on p. 147.
- ▼ **Project model's objects** on the sketch's plane. A geometrical object will be formed in the sketch that will be linked to the model's object directly. Projection of objects is detailed in Section 2.2.2.3.3 on p. 148.



Availability of 3D object choice when working with the sketch depends on filter settings (see Section 2.1.4.1.3 on p. 133). By changing the settings, you can narrow down or expand the range of objects available for specification.

2.2.2.3.1. Snapping to model object when working in a sketch

When creating geometric objects in a sketch, you can use global and local snapping not only to geometric objects of the sketch, but also to model objects. Detailed description of global and local snapping can be found in Section 3.1.1.2 on p. 865. This section describes peculiarities of snapping to model objects.

For the snapping, you can use model objects representing points (vertices, zero points, and so on) or curves (edges, 3D curves, axes, and so on). As a result of snapping, an auxiliary projection of the object, selected for snapping, is formed in the sketch (see Section 2.2.2.3.2 on p. 147), while the created geometric object is parametrically linked to the ghost.

The snap is done as usual: in the process of its creation, position the cursor on the required model object and click the mouse when auxiliary projection of the selected object, snap name and snap point designation (X) appear in the sketch's plane.

For example, if you need to create a segment with the beginning in the middle of the edge's projection, call the **Segment** command, enable global and local snap **Middle** and move the

1. Axes of coordinate systems available in the model can also be used.

cursor to the required edge. Click the mouse as soon as projection of the edge appears in the sketch's plane and X with the *Curve's middle* text appears in the edge's middle.

Keep in mind the following when snapping to model objects.

- ▼ In case of a straight-line object located perpendicular to the sketch's plane, you can only snap to the vertices of the object.
- ▼ It is impossible to create two or more auxiliary projections at the same time. Therefore, it is impossible to make the **Intersection** snap without projecting objects to the sketch whose intersection point is used for snapping.

If necessary, you can disable snapping to all model objects or only to those that are not visible in its current position. For this purpose, the following commands are used: **Snapping to model elements** and **Only to visible model elements** in the **Snaps** button menu (Fig. 2.2.3).

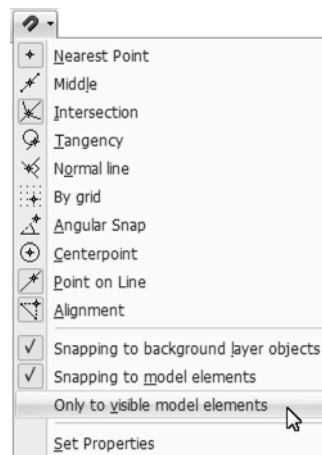


Fig. 2.2.3. Commands for controlling the snap to model objects

2.2.2.3.2. Auxiliary projection of model's object

Construction projection of a model object is created in the sketch automatically when being snapped to the model object and when the model object is used as basic for the sketch object.

Auxiliary project is present in the sketch as long as it takes part in any parametric relations. If the final relation applied to an auxiliary projection is deleted, the projection will be deleted automatically after the work with the sketch is finished.

For example, when specifying the position of the first segment's vertex, snap was made to the center of a round edge. As a result, edge's auxiliary projection (circle or ellipse) has been created in the sketch. The *merged points* relation is formed between its center and vertex. Subsequently, the first vertex of the segment had to be moved to another point and the relation was deleted for this reason. If there are no other relations applied to the construction projection, the projection will be deleted when the sketch mode is exited.



If the parametric mode is disabled in the sketch, or parametrization of snaps is disabled during its setup (see Section 9.2.7.7.1 on p. 1864), no relations are applied during the snap and no auxiliary projections are created.



The user is able to control the auxiliary projection displaying in the sketch. The **Auxiliary projections** button in the **Current state** bar is used for this purpose. If the button is enabled, all auxiliary projections that are present in the sketch will be displayed with dashed lines.



A sketch can contain construction projections right after it is created. This depends on what is used as the sketch plane. Example:

- ▼ a sketch in a flat face contains construction projections of edges of this face,
- ▼ a sketch in a coordinate plane contains construction projection of the origins,
- ▼ a sketch in a plane built by three points contains construction projection of these points,
- ▼ a sketch in a plane built through an edge parallel to another edge contains construction projections of these edges,
- ▼ a sketch in an offset plane does not contain construction projections.

Construction projections that originally existed in the sketch are not deleted from it even if they are not involved in any parametric relations.

2.2.2.3.3. Projecting model's objects to sketch

Sometimes, lines or points need to be created in the sketch to represent 3D object projections to the sketch's plane.



To create a projection of an object, apply the **Operations — Project Object** command. Button to apply this command is located on **Geometry** panel.

Specify the object the projection of which should be obtained — a vertex, face, edge, 3D curve, axis, point and so on.

Projection of the specified object is created in the sketch.

Depending on which object is selected for projecting, only one of the following geometric objects can be created in the sketch:

- ▼ point with the *Auxiliary* style, if a vertex is projected,
- ▼ auxiliary line, if coordinate or auxiliary axis is projected,
- ▼ object of the **projection curve** type of the *Basic* style, if an edge, 3D curve or face is projected; in the latter case, projection curves are created (edge projections) that limit the face.



If a straight-line edge perpendicular to the sketch's plane is selected for projecting, one of the edge's vertices will be projected to the sketch.

A parametric relation (the *projection relation*) is automatically applied to the created projection. This ensures permanent link of the projection to the source 3D object.



Removal of the *projection link* relation results in removal of the link between the projection and source object. Projection becomes an ordinary geometrical object which can be moved or edited. The result will be the same if the **Explode** command is applied to the projection.

Projection of end vertex relations are also applied to the projection curve, if it is open. These relations ensure coincidence of outermost vertices of the projection curve with projections of outermost vertices of the source object.

Several different object projections can be built in a sketch in a single call of the **Project object** command. For this purpose, they should be specified one by one.



To finish projecting, click the **Interrupt command** button in the Special Control Panel.

Working with the projection curve

- ▼ It is impossible to change the projection curve's form (for example, it is impossible to change a circle's arc radius). However, the curve can be trimmed. Projection curve can also be used as a restricting object in the alignment or extension commands.
- ▼ If a projection curve is cut in several sections, all other sections will be selected, if one of them is selected.
- ▼ The length of the projection curve can be modified by moving handles on its ends (or ends of its parts). To move an end point of the curve, delete the *projection of end vertex* link laid on it. A point can only be moved along the curve. If the projection curve is a segment or arc of a circle/ellipse, handles can be moved along the curve's extension (i.e. the curve can be made longer than the projection of the original object).
- ▼ When copying a projection curve, a normal geometrical object (or objects) is created.
- ▼ Editing commands can be applied to the projection curve, if the mode, in which source objects remain after the command execution, is enabled in them. As a result of these commands, an ordinary geometrical object (or objects) is created.
- ▼ In order to restore the initial state of the projection (after section, for example), use the **Restore curve** command in the context menu. The command is available in the context menu as long as the curve maintains a link to the source object (i.e. there is a *projection link* in the list of constraints imposed on it).

2.2.2.4. Peculiarities of using some type of objects in a sketch

2.2.2.4.1. Polygon, rectangle, polyline

Polygons, rectangles, polylines created in a sketch are automatically split to separate segments (they do not remain single objects as in the event of working with graphic documents). In the parametric mode, relations and limitations are applied to these segments. For example, *horizontality* and *verticality* limitations are applied to a rectangle's sides. *Coincidence* relations are applied to their vertices.



Polygons, rectangles, polylines copied to a sketch from graphic documents are not split to segments.

2.2.2.4.2. Text

Sketch may contain text. After leaving the sketch mode, all text contained it will be automatically transformed into one or several contours that consist of spline curves.

Sometimes, automatic transformation may result in contours that do not meet the requirements of an operation (tangent, for example). In such cases, you can try to change the text's font. If a contour still does not fit an operation, you can apply the **Transform into NURBS** command in the sketch. The resulting lines will not be text (i.e. text must be entered and trans-



formed into NURBS again in order to edit it). However, they will follow the letters' contours, and you will be able to correct the sketch so that it meets the operation's requirements.



Text after NURBS transformation will have to be deleted from the sketch or moved to its hidden layer.

2.2.2.4.3. Insertion of fragments, macro elements

The following sketch objects **cannot be used** to perform operations:

- ▼ macro elements inserted from libraries or copied through the clipboard from graphic documents,
- ▼ fragments inserted from libraries,
- ▼ insertions of local or external fragments made using the **By external link** or **Embed in document** method.

Fragments and macro elements must be split into separate geometric objects after insertion to the sketch using the **Editor — Explode** command.

2.2.2.4.4. Technological designations, tables, hatches

It is impossible to create technological designations, tables, hatches and some other objects in sketches.

If some of the objects, that cannot be created in a sketch, appear in the sketch (for example, copied from a graphic document), you can keep working without problems. These objects will just be disregarded when creating a three-dimensional feature. They are saved in the sketch. They can be reviewed while editing the sketch.

2.3. Bodies

2.3.1. General Information on Bodies

You can create a new body in a model using one of the following operations:

- ▼ **Extrusion.** Creates a body by sweeping a section along a linear path a given distance (see Section 2.3.2 on p. 152).
- ▼ **Rotation.** Creates a body by rotating a section around an axis with a specified angle (see Section 2.3.3 on p. 166).
- ▼ **By Sections.** Creates a body by connecting several sections (see Section 2.3.5 on p. 177).
- ▼ **Sweep Feature.** Creates a body by sweeping a section along any given path (see Section 2.3.4 on p. 173).
- ▼ **Thickening.** Creates a body by adding a material layer to a specified surface (see Section 2.3.6 on p. 184).
- ▼ **Sheet Body.** Creates a special body type — a sheet body (see Section 2.4 on p. 231).
- ▼ **Surfaces Knitting.** Creates a body surrounded by the specified surfaces (see Section 2.6.2.4 on p. 478).

Once the new body has been created, additional volumes are “attached” to it or “cut” out of it, i.e. the body's material is being added or removed (Fig.2.3.1).

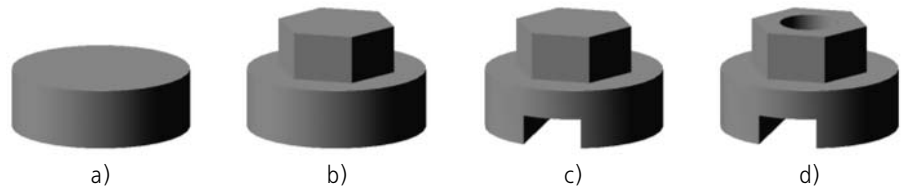


Fig. 2.3.1. Constructing a body:
a) new body— cylinder; b) prism adding;
c) “cut” of a prism; d) “cut” of a cylinder

Examples of removing material of a part are various holes, slots, or grooves, while examples of adding volumes are bosses, protrusions, and edges (Fig. 2.3.2).

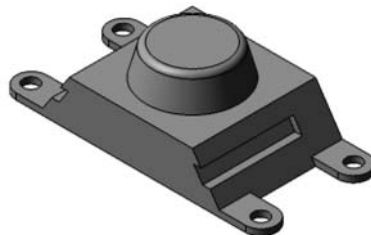


Fig. 2.3.2. The boss and the feet are glued to the body whereas the slots and holes are cut from it.

Adding and deleting extrusion, revolution, lofted, swept features are described in Sections 2.3.2–2.3.4; adding a material layer to the body faces is described in Section 2.3.6 on p. 184. Other options for body editing (adding chamfers, fillets, edges, creating shell, etc.) are described in Section 2.3.8 on p. 187.

A model may consist of multiple bodies. Boolean operations can be performed with bodies (see Section 2.3.7 on p. 185).

Using bodies that are present in the model, you can create new ones using array building commands, see Section 2.7 on p. 509.

2.3.1.1. How to start creating a model

Before creating a model, user always faces the question how to start creating. To answer this question, you need to have an approximate idea of the future part design.

Conceptualize the structure from chamfers, fillets, grooves and other small structural features. Divide the part into constituent features (parallelepipeds, prisms, cylinders, cones, tori, swept features etc.).

Most often, the largest of those features is created first. If the structure of the part has several features of comparable sizes, you can start construction with any of them.



Sometimes, it may be rational to construct several bodies and then combine them to create such a part. Some parts, for example, segmented washers, are made of several bodies not connected to each other. Multibody modelling is detailed in section 2.3.12 on p. 228.

Sometimes, a simple feature circumscribing the part to be designed (or its portion) is used as the base (i.e., a block or cylinder).

In some cases, the selection of the first element (and the order of further part design) may be done based on the manufacturing technology involved.

Generally speaking, there are no universal instructions for creating three-dimensional models. Designers gain experience in a preferred modelling approach after independently creating several models.

2.3.2. “Extrude” and “Cut by extrusion” operations

2.3.2.1. Extrusion Feature

Extrusion feature is created by sweeping section along the straight-line guide in either or both sides at the specified distance. For example, a feature shown in Fig. 2.3.3, is created by extruding a sketch in the direction perpendicular to its plane.

Extruded element may be an individual body or may be attached to a body or cut out of it.

Operation **Extrude** serves to create a new body of extrusion or to add an extrusion feature to an existing body (i.e. to add a material), operation **Cut-Extrude**— to cut an extrusion feature from a body (to remove material).

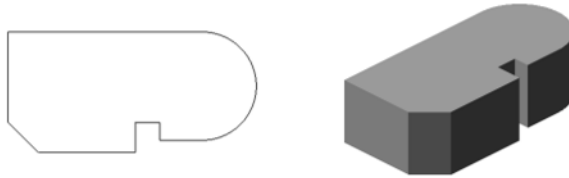


Fig. 2.3.3. Sketch and element produced by the extrusion operation

A face, sketch, edge or 3D curve (including contour constructed by sketch lines, by face edges or arbitrary¹) may be used as a section of extrusion feature.

During extrusion of face, closed sketch, closed **Contour on Face** or **Sketch Contour**, the choice between thin wall and body feature is available. (Fig.2.3.4). If section is open, only thin wall feature construction is available.

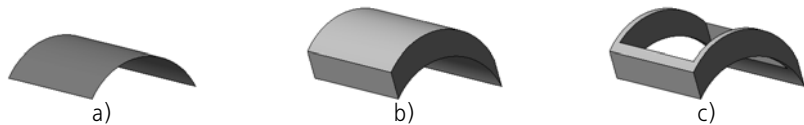


Fig. 2.3.4. Face extrusion: a) section (cylindrical face), b) body feature, c) thin wall feature

If section is a flat face, sketch, contour constructed by sketch lines or flat face, and it is extruded at the direction perpendicular to it, sloping to side faces of feature is possible (Fig.2.3.5).



Fig. 2.3.5. Slope of side faces of extrusion feature:
a)

During edge or 3D curve extrusion (including contour of **Arbitrary** type), only thin wall feature construction is possible. Slope of side faces is impossible. Example of 3D curve extrusion is shown in the Figure2.3.6.

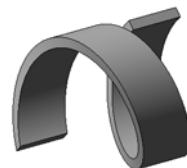


Fig. 2.3.6. Extrusion element with a spiral section

1. It means that has **Contour on Face**, **Sketch Contour** and **Arbitrary Contour** type, see Section 2.5.4.24.1 on p. 438.

2.3.2.2. Execution of the operation



1. Start an operation. To do this, complete the following steps:

▼ click the required button on the **Edit Part** or **Edit Model** panel.



▼ or call the required command from **Operations** menu.

Availability condition of **Cut-Extrude** command: model including a body or a component.

After operation launch, tabs with operation parameters controls appear on the Properties bar.

2. Specify the operation parameters:

▼ **Section** (see Section 2.3.2.3).

▼ **Extrusion Direction** (see Section 2.3.2.4).

▼ **Extrusion Depth** (see Section 2.3.2.5).

▼ **Inclination Angle** (see Section 2.3.2.6).

▼ **Thin Wall Settings** (see Section 2.3.2.7),

▼ **Result** (see Section 2.3.2.8).

All parameter values, once entered or modified, are immediately displayed on the screen as the ghost of the extruded feature. A ghost allows a visual control over parameter accuracy.



To set up such parameters as slope angle and wall thickness, extrusion direction and depth, you can use handles (handles are detailed in Section 2.1.4.2.2 on p. 138).



3. Click **Create Object** button to complete the operation.

4. If such operation results in a body comprising several parts, the Modify set of parts process will be started automatically after the end of the operation. Select the parts to be hold (see Section 2.3.11.1).

Operation result — new body, pasted or cut element — appears in the graphic area.

The operation's icon appears in the Construction Tree. It coincides with icon on the button of command which starts operation. Exception is an icon of new extrusion body— it differs from an icon on the button.



an icon of new extrusion body

2.3.2.3. Extrusion feature section

2.3.2.3.1. Dependence of the result of extrusion operation of the section

In Table 2.3.1, you can see peculiarities of the extrusion operation's performance and results depending on the choice of section.

Tab. 2.3.1. Peculiarities of extrusion operation execution, depending on an object used as a section

Object	Peculiarities of operation execution and result
sketch, contour created by sketch lines	<ul style="list-style-type: none"> ▼ if section is closed, you can create both thin wall and body feature ▼ slope of feature side faces is possible ▼ sketch must meet the requirements presented in Section 2.3.2.3.3
face, contour created by face edges	<ul style="list-style-type: none"> ▼ you can create both thin wall and body feature (if a closed contour section is used) ▼ if a face is planar, slope of feature side faces is possible
edge, 3D curve, arbitrary contour	<ul style="list-style-type: none"> ▼ only thin wall feature can be created ▼ slope of feature side faces is impossible

2.3.2.3.2. Specification of extrusion feature section

You can use an existing object in the model as a section of the extrusion element— an edge, curve, sketch or face, or you can create a new object— contour— without interrupting the operation.



To select the existing object as a section, specify it in the Design Tree or in the graphic area.

To create a contour, click the **Contour** button on the Advanced control panel. Contour creation process will be started (see Section 2.5.4.24 on p. 437). Specify objects included into contour and click **Create Object** button. The system will resume the extrusion operation process, created contour will appear in the Design Tree and will be automatically selected as a extrusion feature section.

Name of the section object shall appear in the **Section** field on Properties toolbar.



If, when moving section in the preset direction, two or more section points have the same paths, it will not be possible to complete the operation. You need to change section or direction.



Extrusion feature section can be specified before the operation launch.



You can change extrusion feature section at any time. To do this, activate **Section** switch and then specify the desired object or construct a contour.



A sketch selected as an extrusion feature section can be edited without interrupting the operation. To do this, click the **Edit Sketch** button on the Special Control Panel. The system will switch to the sketch editing mode. Make the necessary modification in the sketch. Then click **Sketch** button on the **Current status** panel or click the sketch mode icon in the document window to return to the extrusion operation process.



2.3.2.3.3. Extrusion Feature Sketch Requirements

- ▼ Sketch objects can be a part of one or several contours.
- ▼ If there are several contours, they all should be either closed or open.
- ▼ If the contours are closed, they can be nested. The level of nesting is not limited.

2.3.2.4. Direction of Extrusion

2.3.2.4.1. Guide Object

If the feature section is a sketch or flat face, this object is automatically selected as a guide. If section is contour created by sketch or by flat face edges, a corresponding sketch or face is automatically selected as a guide object. If necessary, you can change guide object.

You can use the following guide objects for extrusion:

- ▼ An existing planar or straight-line object (the list of objects is provided in Table 2.1.7 on p. 131).
 - ▼ Straight-line object defines direction parallel to itself.
 - ▼ Planar object defines direction perpendicular to its plane.
- ▼ Vector.



To select an existing object as a guide, activate **Guide Object** switch on the Properties bar and then specify the required object in the Design Tree or in the graphic area.





To create a vector, click the **Vector Drawing** button on the Advanced control panel. For more details about drawing vectors, see Section 2.8.5 on p. 613.

The name of the guide object is displayed in the **Guide Object**.



2.3.2.4.2. Selecting the extrusion direction

The corresponding direction of extrusion can be selected from the **Direction** list (see Table 2.3.2) in the Property bar.

Tab. 2.3.2. Available directions of extrusion

	Value of Direction option	Direction of extrusion. Specifics of defining extrusion depth
	Forward	In the forward direction relative to the section* . When choosing this option on the property bar, there is the Method 1 list to select a method of setting the extrusion depth (see Section2.3.2.5).
	Reverse	In the reverse direction relative to the section* . When choosing this option on the property bar, there is the Method 2 list to select a method of setting the extrusion depth (see Section2.3.2.5).

Tab. 2.3.2. Available directions of extrusion

	Value of Direction option	Direction of extrusion. Specifics of defining extrusion depth option
	Two directions	In both directions relative to the section. When choosing this option on the property bar, there are Method 1 and Method 2 lists to select a method of setting the extrusion depth in each direction (see Section 2.3.2.5).
	Middle plane	To both directions symmetrically relative to the section. If this option is selected, Method 1 and Method 2 lists are unavailable, for only one method to define the distance, i.e., the exact setting of overall extrusion depth, is available. Enter the required value into the Distance 1 .

* To make out directions (forward and reverse), the ghost in the detail viewing window has an arrow indicating the forward direction. If **Forward Direction** option is selected, extrusion shall be done in the direction of the arrow. If **Reverse Direction** option is selected, the revolution shall be done against the arrow.

2.3.2.5. Depth of Extrusion

Once the direction is selected, you need to specify the distance extrusion value. To do this, select the desired option from **Method 1** (and/or **Method 2** — depending on the selected direction) list on the Properties bar. The list of methods of selecting the extrusion depth has the following options (see below):



▼ **At Distance,**



▼ **Through All,**



▼ **Up to Vertex,**



▼ **Up to Surface,**



▼ **Up to Nearest Surface.**

2.3.2.5.1. Extrusion at the specified distance

Extrusion is done exactly for the distance specified in **Distance** field.

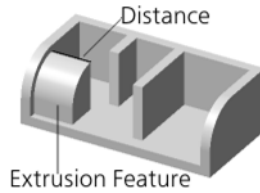


Fig. 2.3.7. Extruding by a distance

2.3.2.5.2. Extrusion through all

Depth of extrusion is defined automatically.

The element is extruded before it reaches the plane that goes perpendicular to the extrusion direction through the dimensional 3D cube's outermost point from the section in the extrusion direction.

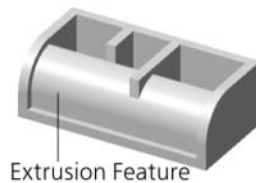


Fig. 2.3.8. Extrusion through all

When creating a circumscribing block, only bodies are taken into account, therefore, if there is no bodies in model, **Through All** method of depth definition is unavailable.

2.3.2.5.3. Extrusion up to vertex

Depth of extrusion is defined automatically by position of a vertex selected by user.

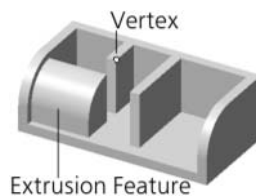


Fig. 2.3.9. Extrusion up to vertex

- ▼ If a face, sketch, or contour built by sketch lines or face's edges, are used as a section, the depth is calculated so that the element's endface (or its continuation) goes through the specified vertex.
- ▼ If an edge, 3D curve, or arbitrary contour are used as a section, the extrusion depth is calculated as the distance between the specified vertex and the point nearest to the section which lies on the straight line going through the vertex in the extrusion direction.



To specify a vertex defining extrusion depth, activate **Vertex** switch on the Properties bar and select the desired point object in a graphic area (point can be also specified in the Design Tree). The name of the selected object is displayed in a field beside the switch.

Using the **Cut** option, you can cut the element with a plane going through the specified vertex perpendicular to the extrusion direction. In this case, the element will have a flat endface irrespective of the section form.

If necessary, you can set the distance at which the element's endface should be positioned from the vertex, i.e. increase or decrease the automatically defined extrusion depth (see Section 2.3.2.5.6 on p. 160).



Extension of complex shape faces is not always possible. Therefore, if the specified vertex is positioned in such a way that, provided the extrusion direction is set, the element's endface does not go through the vertex, in some cases, construction will only be possible if the **Cut** option is enabled.

2.3.2.5.4. Extrusion up to surface

Depth of extrusion is defined automatically by position of a face or a plane specified by User.

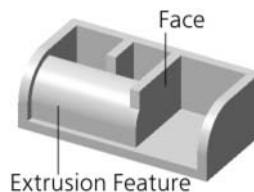


Fig. 2.3.10. Extrusion up to surface



To specify a surface defining extrusion depth, activate the **Face** switch on the Properties bar and select the desired object in a graphic area. The name of the selected object is displayed in a field beside the switch.

The element is extruded up to the selected surface. The endface of the feature has the surface form.

If necessary, you can set the distance to the element's endface from the surface (see Section 2.3.2.5.6 on p. 160).

It is recommended to choose section, direction, and surface so that the surface fully intersects the extrusion element. Otherwise, the construction is done only in case of selected surface can be extended. Extension of complex shape surfaces is not always possible.

2.3.2.5.5. Extrusion up to nearest surface

Depth of extrusion is defined automatically. The feature is extruded precisely to the nearest faces of the part in the extrusion direction (in other words, until the extrusion will meet a face (or faces) on its way). The endface of the feature has the face (faces) form.

If there are no bodies in a model, **Up to Nearest Surface** method of depth definition is impossible.

It is recommended to choose section and direction so that the extrusion element intersects bodies existing in the model.

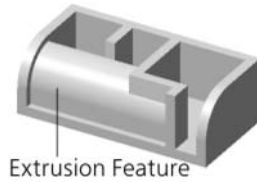


Fig. 2.3.11. Extrusion up to nearest surface



Method of defining the **Up to Nearest Surface** depth is useful for extrusion of an feature up to a stepwise or curvilinear edge (see Fig.2.3.12).



Fig. 2.3.12. Extrusion of a feature up to the nearest curvilinear surface

2.3.2.5.6. Setting distance to the object

If **To Vertex** or **To Surface** method of creating was selected, then, after vertex or surface specifying, you need to specify distance to it.

To do that, enter required distance between end face of the feature and the object into the **Distance** field.

If the object is to be extruded exactly up to the object, enter zero distance.



If the distance to the object is nonzero, it can be applied either in the extrusion direction (in this case, the feature will be extruded “overshooting” the object by the specified distance), or counter to the extrusion direction (in this case, the feature “undershoots” the object by the specified distance). To specify direction of measuring the distance to the object, activate **Before Object** or **Behind Object** switch in the **Type1** group.

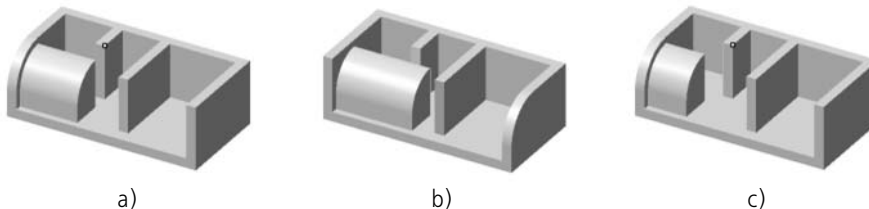


Fig. 2.3.13. Extrusion of feature:
a) exactly to the vertex; b) behind the vertex; c) close to the vertex



In case of extrusion to the surface, the equidistant surface is created at a preset distance from it, up to which the extrusion is made.

2.3.2.6. Angle of Slope

Slope of side faces of extrusion feature is possible if the following conditions are satisfied:

- ▼ If a flat face, sketch, contour constructed by sketch lines or flat face edges is used as a feature section,
- ▼ extrusion direction is perpendicular to section plane.



To extrude in the forward or reverse direction (Fig. 2.3.14), specify the direction of the slope by activating the **Inward** or **Outward** switch in the **Slope** group. Enter the inclination angle value in the **Angle** field.



To extrude in both directions, the direction and the slope value are required to be input twice, for both forward and reverse directions (Fig.2.3.15).

To extrude from midplane, the slope parameters are specified once and considered equal in both directions.

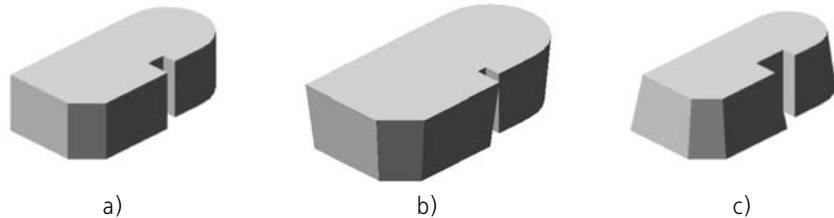


Fig. 2.3.14. Extrusion in one direction:
a) without slope, b) slope outward, c) slope inward

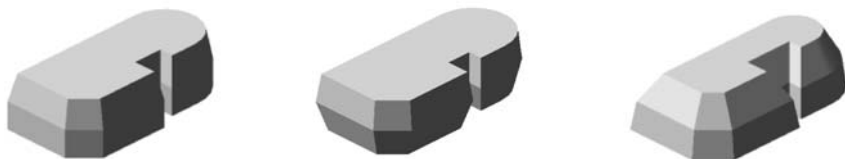


Fig. 2.3.15. Extrusion in both directions with different slope parameters

2.3.2.7. Thin Wall

You can create thin wall shell during creation of extrusion, revolution, lofted, swept features and some other objects.

When a shell is formed, a material layer is added to the feature surface (without its "end edges").








If construction of a thin wall shell based on a complex body surface (not a feature) is required, use the **Thin Wall Shell** command (see Section 2.3.8.3 on p. 196).

Options to control the construction of a thin wall feature are available during feature parameter setting. They are located on Properties toolbar **Thin Wall** tab.

Specify the direction of adding a material layer relative to the surface. To do this, select the required option in **Thin Wall Construction Type** list (see Table 2.3.3).

Tab. 2.3.3. Possible options for constructing a thin wall

	Value of Thin Wall Type option	Direction of adding a material layer. Specifics of defining wall thickness
	No	The thin wall is not created, i.e. body element is constructed. This option is available, if the section is a closed sketch (when performing rotation and extrusion operations, construction of a body feature is also possible using other objects as the section, see Sections 2.3.2.3.1 on p. 154 and 2.3.3.3.1 on p. 168).
	Outside	In the forward direction relative to the surface [*] . Enter the thickness in the Wall Thickness 1 field. Only positive values are allowed.
	Inside	In the backward direction relative to the surface [*] . Enter the thickness in the Wall Thickness 2 field. Only positive values are allowed.
	Two directions	In both directions relative to the surface. Enter in the Thickness of Wall 1 field thickness in the outward direction, in the Thickness of Wall 2 field— thickness in the inward direction (Fig. 2.3.16, a). One of the values may be negative. Negative thickness is calculated in the direction which is opposite to the positive one. This means removal of the material. Therefore, you can construct a thin wall at a distance from the element's surface (Fig. 2.3.16, b).
	Middle plane	In both directions symmetrically relative to the surface. Enter the overall thickness in the Wall Thickness 1 field. Only positive values are allowed.

* Use a ghost for visual control to direct adding of material.

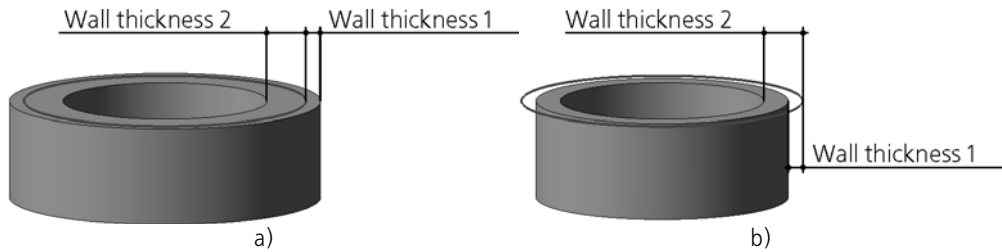


Fig. 2.3.16. Construction of thin wall in both directions (on an extrusion feature example; section is shown with a thickened line): a) wall 1 thickness and wall 2 thickness are positive; b) thickness of wall 2 has not changed, while thickness of wall 1 has been attributed the opposite sign

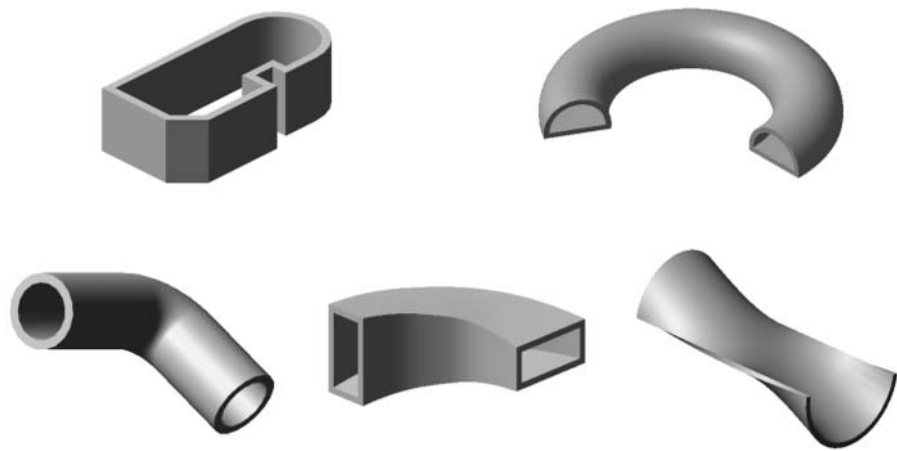


Fig. 2.3.17. Examples of thin wall features



Word "thin" in the term "thin wall" is mostly a relative term. In some cases, you can add to the surface created by the contour movement a layer of material with such thickness that largely exceeds linear sizes of the contour in the sketch. The resulting feature will not be "thin-layer" in the direct sense of the word. However, in most cases, the thickness of the created feature is one or even several times less than its external dimensions, and this is why the wall is called a "thin" one.



For side walls of the thin wall extrusion feature with a slope, the specified thickness shall be complied in the section plane and in parallel planes.

2.3.2.8. Operation result

The result of the operation is:

- ▼ When adding material: **an individual body** or a **feature attached** to an existing body (bodies). When attaching, you can specify operation application area—a set of bodies with which a new element shall be merged.
- ▼ When removing material: **feature subtraction** from body (bodies) or **feature intersection** with body (bodies). When subtracting or intersecting, you can set the area to which the operation will be applied—a set of bodies and/or components whose material is to be removed as a result of the operation.

Operation result selection is described below, setting of operation application area—in section 2.3.10 on p. 217.

2.3.2.8.1. Result of adding material

When performing extrusion, rotation, by sections, sweeping, as well as when creating a sheet body and adding thickness, you can specify if the element should be a separate body or it should be joined with existing ones (i.e. to attach them to it).

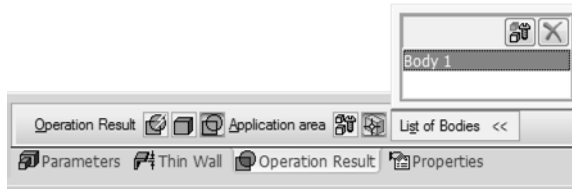


Fig. 2.3.18. Selecting the result of adding material operation

To choose adding material option, go to the **Operation Result** tab of the Properties bar (Fig.2.3.18) and activate the desired switch in the **Operation Result** group.



- ▼ **Auto Union**— operation result is defined automatically. In this case, the construction is done as follows:
 - ▼ If the feature being created does not intersect and has no common surfaces with any of the existing bodies, then a new body is created,
 - ▼ If the created feature intersects or has a common surface with some of the existing bodies, then the feature is merged with those bodies.



- ▼ Activate the **New Body** switch to have the feature being created as a separate body regardless of whether it intersects other bodies or not.



- ▼ Activate the **Merge** switch to select arbitrary bodies to merge with the element being created. You can use the **Application Area** group of switches to select bodies (see Section 2.3.10.3.3 on p. 221). If an element is joined with one body, the total number of bodies in the model will not change. If the element is joined with several objects, the total number will decrease, since all joint bodies become parts of a new body.

By default, the **Auto Union** switch in **Operation Result** group is active. Note that this switch is only available when an operation is being performed. To verify this, enter the editing mode of the feature created with the **Auto Determination** switch enabled. You will see, that depending on operation result, **New Body** or **Union** switch is active on the Properties bar. In the latter case, the **List of Bodies** containing names of bodies with which there are intersections will appear on the Properties bar.

2.3.2.8.2. Result of material removing

When cutting a feature of extrusion, revolved, loft, or sweep type out of a model, material can be deleted inside or outside the feature surface, that is, the result can be the model less the feature or the model intersection with the feature (see diagram in Fig.2.3.19).

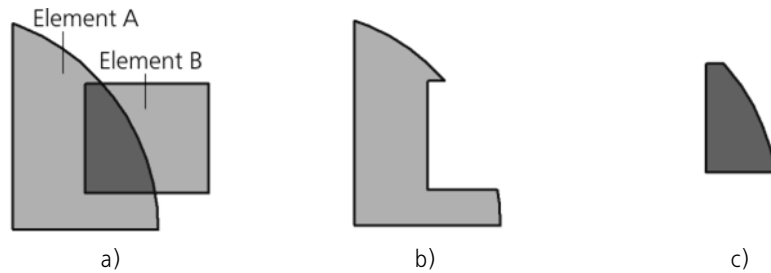


Fig. 2.3.19. Different methods of cutting: a) initial features, b) result of subtracting feature B from feature A, c) result of intersecting between features A and B

To select a cutting option, activate the **Cut** Properties toolbar tab (Fig. 2.3.20).

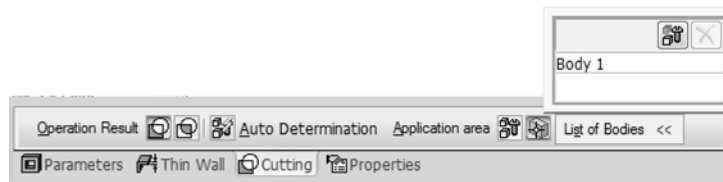


Fig. 2.3.20. Selecting the result of removing material operation



Activate the switch corresponding to the required method of cutting in the **Operation Result** group, i.e. **Subtraction of Feature** or **Intersection of Features**.

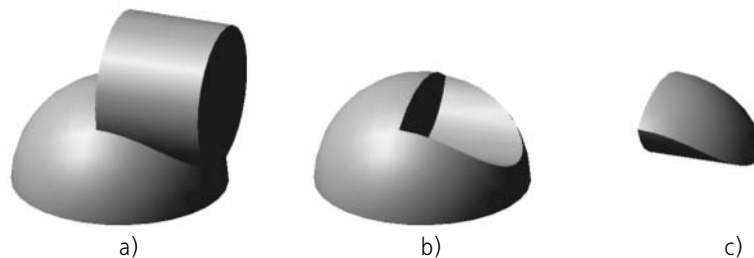


Fig. 2.3.21. Different methods of cutting: a) initial features, b) result of subtraction of cylinder from hemisphere, c) intersection of cylinder and hemisphere

2.3.3. "Rotation" and "Cut with rotation" operations

2.3.3.1. Revolution Feature

Revolution feature is created by rotation of section about the axis in either one or both sides at the specified angle. For example, a feature shown in Fig. 2.3.22, is created by rotating a sketch about an axis lying in its plane at 270°.



Fig. 2.3.22. Sketch, axis and element produced by the revolved operation

Rotation element may be an individual body or may be attached to a body or cut out of it.

Operation **Revolve** serves to create a new body of revolution or to add a revolution feature to an existing body (i.e. to add a material), operation **Cut-Revolve**— to cut a revolution feature from a body (to remove material).

A face, sketch, edge or 3D curve (including contour constructed by sketch lines, by face edges or arbitrary¹) may be used as a section of revolution feature.

During revolution of face, closed sketch, closed **Contour on Face** or **Sketch Contour**, the choice between thin wall and body feature is available. (Fig.2.3.23). If section is open, only thin wall feature construction is available.

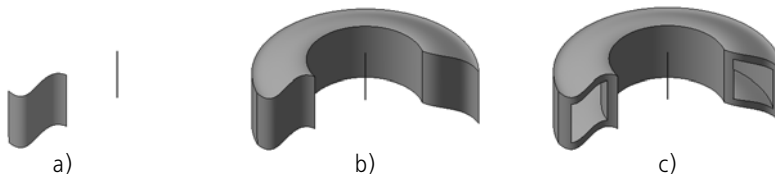


Fig. 2.3.23. Face Rotation: a) section and axis, b) body feature, c) thin wall feature

If the section is an open sketch or an open contour built by the sketch lines or flat face's edges and rotates around the axis within the section's plane, the construction type selection is available— toroid or spheroid (Fig.2.3.5). When building a spheroid, a choice between body and thin wall feature is available. Toroid may be thin wall only.

During edge or 3D curve revolution (including contour of **Arbitrary** type), only thin wall feature construction is possible. Example of 3D curve revolution is shown in the Figure 2.3.6.

1. It means that has **Contour on Face**, **Sketch Contour** and **Arbitrary Contour** type, see Section 2.5.4.24.1 on p. 438.

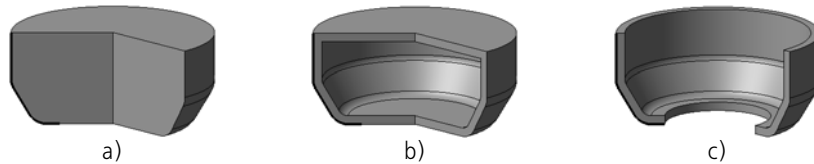


Fig. 2.3.24. Spheroid and toroid construction (section— sketch— marked with a thickened line):
a) a solid spheroid, b) a thin-wall spheroid, c) a toroid



Fig. 2.3.25. Rotation element with a spiral section

2.3.3.2. Execution of the operation



1. Start an operation. To do this, complete the following steps:

▼ click the required button on the **Edit Part** or **Edit Model** panel.



▼ or call the required command from **Operations** menu.

Availability condition of **Cut-Rotate** command: model including a body or a component.

After operation launch, tabs with operation parameters controls appear on the Properties bar.

2. Specify the operation parameters:

▼ **Section** (see Section 2.3.3.3).

▼ **revolution axis** (see Section 2.3.3.4);

▼ **creation type** (see Section 2.3.3.5);

▼ **revolution direction** (see Section 2.3.3.6);

▼ **revolution angle** (see Section 2.3.3.7).

▼ **Thin Wall Settings** (see Section 2.3.2.7),

▼ **Result** (see Section 2.3.2.8).

All parameter values, once entered or modified, are immediately displayed on the screen as the ghost of the extruded feature. A ghost allows a visual control over parameter accuracy.



You can use handles to specify rotation angle and wall thickness (for more on handle see Section 2.1.4.2.2 on p. 138).



3. Click **Create Object** button to complete the operation.

4. If such operation results in a body comprising several parts, the Modify set of parts process will be started automatically after the end of the operation. Select the parts to be hold (see Section 2.3.11.1 on p. 226).

Operation result — new body, pasted or cut element — appears in the graphic area.

The operation's icon will appear in the Design Tree matching the icon on the command's button which is used to run the operation. Exception is an icon of new revolution body—it differs from an icon on the button.



— an icon of new revolution body

2.3.3.3. Section of revolution element

2.3.3.3.1. Dependence of the result of revolution operation of the section

In Table 2.3.4, you can see peculiarities of the revolution operation's performance and results depending on the choice of section.

Tab. 2.3.4. Peculiarities of revolution operation depending on an object used as a section

Object	Peculiarities of operation execution and result
sketch, face, contour constructed by sketch lines or by face edges	<ul style="list-style-type: none"> ▼ Construction Type Selection — toroid or spheroid (see Section 2.3.3.5 on p. 170) — is available if the following conditions are satisfied: <ul style="list-style-type: none"> ▼ a section is open and represents a sketch or contour constructed by sketch lines or by flat face edges, ▼ revolution axis lies in the section plane. ▼ If section is closed, you can create both thin wall and body feature. If section is open, you can choose between a thin-wall or solid element when creating a spheroid; in other cases, a thin-wall element will be constructed when the section is open. ▼ Sketch must satisfy the requirements presented in Section 2.3.3.3.3.
edge, 3D curve, arbitrary contour	<ul style="list-style-type: none"> ▼ only thin wall feature can be created ▼ unavailable to select type of construction— toroid or spheroid

2.3.3.3.2. Setting of the Revolution Feature Section

You can use an existing object in the model as a section of the rotation element— an edge, curve, sketch or face, or you can create a new object — contour — without interrupting the operation.



To select the existing object as a section, specify it in the Design Tree or in the graphic area.

To create a contour, click the **Contour** button on the Advanced control panel. Contour creation process will be started (see Section 2.5.4.24 on p. 437). Specify objects included into contour and click **Create Object** button. The system will resume the revolution operation process, created contour will appear in the Design Tree and will be automatically selected as a revolution feature section.

Name of the section object shall appear in the **Section** field on Properties toolbar.



If, when rotating section around the chosen axis, two or more section points have the same paths, it will not be possible to complete the operation. You need to change section or axis.



Revolution feature section can be specified before the operation launch.



You can change revolution feature section at any time. To do this, activate **Section** switch and then specify the desired objects or construct a contour.



A sketch selected as an revolution feature section can be edited without interrupting the operation. To do this, click the **Edit Sketch** button on the Special Control Panel. The system will switch to the sketch editing mode. Make the necessary modification in the sketch. Then click **Sketch** button on the **Current status** panel or click the sketch mode icon in the document window to return to the rotation operation process.



2.3.3.3. Revolution Feature Sketch Requirements

- ▼ Sketch objects can be a part of one or several contours.
- ▼ If there are several contours, they all should be either closed or open.
- ▼ If the contours are closed, they can be nested. The level of nesting is not limited.

Besides the section, sketch may also contain the rotation axis. In this case, additional requirements shall be met:
- ▼ the axis should be drawn in a sketch as a line segment with *Centerline* line style or as an *Centerline* object type,
- ▼ no contour should intersect the axis (or its extension),
- ▼ some contour points may be on the axis (or its extension),
 - ▼ for closed contours – any of their points and/or segments,
 - ▼ for open contours – only their outermost points.

2.3.3.4. Revolution Axis,

To specify revolution axis, you can:

- ▼ select an existing object:
 - ▼ straight-line object (edge or axis) — the object will be used as axis,
 - ▼ rotation surface (except for the sphere) — the surface rotation axis will be used,
- ▼ construct auxiliary axis.

The axis of revolution may be defined automatically. For this purpose, it should be present in the same sketch with the element's section and be a segment with the *Axial* type or the *Axial line* object. If necessary, you can select another axis.



To select a straight-line object or revolution surface, activate **Axis** switch on the Properties bar and then specify the required object in the Design Tree or in the graphic area.



To create an axis, click the **Axis through Vertex to Object** button on the Special Control Panel. Axis creation process will be started (see Section 2.8.1.5 on p. 588). Create the axis and click the **Create Object** button. The system will resume the revolution operation process, created axis will appear in the Design Tree and will be automatically selected as a revolution axis. The name of the axis-setting object will appear in the **Axis** field.




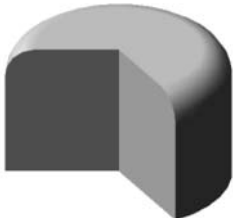
2.3.3.5. Construction type

Construction Type Selection — toroid or spheroid — is available, if:

- ▼ a section is open and represents a sketch or contour constructed by sketch lines or by flat face edges,
- ▼ revolution axis lies in the section plane.

To select the right building type, activate the applicable switch in the **Method** group (see Table 2.3.5).

Tab. 2.3.5. Types of revolution feature

Value of Type option	Aspects of feature construction	Construction result
 Toroid	<p>When building a toroid, only the section itself is revolved. A layer of material is added to the generated surface. Therefore, a thin-wall shell appears, i.e. a feature with a hole along the axis of revolution. Setting of a thin wall is detailed in Section 2.3.2.7.</p>	
 Spheroid	<p>When building a spheroid, the section end points are projected onto revolution axis. The feature is then built considering these projections. Consequently, you get a body feature.</p>	



If you need to build a rotation element with flat edges (Fig. 2.3.26), create a section with the profile of this element and activate the **Spheroid** switch when performing the operation.

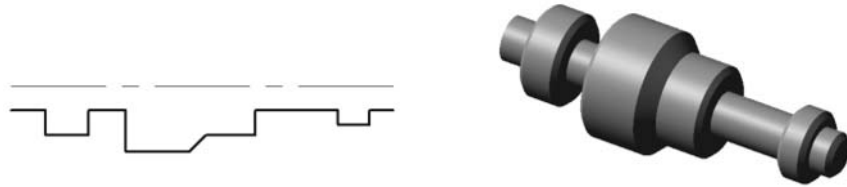






Fig. 2.3.26. Construction of a revolution feature with plane end edges

2.3.3.6. Direction of Revolution

Direction of revolution can be selected from the **Direction** list (see Table 2.3.6).

Tab. 2.3.6. Available directions of extrusion

	Value of Direction option	Direction of the revolution. Aspects of revolution angle setting
	Forward	In the forward direction relative to the section*. When choosing this option on the property bar, there is the Method 1 list to select a method of setting the rotation angle (see Section 2.3.3.7).
	Reverse	In the reverse direction relative to the section*. When choosing this option on the property bar, there is the Method 2 list to select a method of setting the rotation angle (see Section 2.3.3.7).
	Two directions	In both directions relative to the section. When choosing this option on the property bar, there are Method 1 and Method 2 lists to select a method of setting the rotation angle in each direction (see Section 2.3.3.7).
	Middle plane	To both directions symmetrically relative to the section. If this option is selected, Method 1 and Method 2 lists are unavailable, for only one method to define the angle, i.e., the exact setting of overall rotation angle, is available. Enter the required value into the Angle 1 .

* For distinguishing directions (forward and reverse), the ghost in the detail viewing window has an arrow indicating the forward direction. If **Forward Direction** option is selected, revolution shall be done in the direction of the arrow. If **Reverse Direction** option is selected, the revolution shall be done against the arrow.

2.3.3.7. Rotation Angle

Once the direction is selected, you need to specify the angle revolution value. To do this, select the desired option from **Method 1** (and/or **Method 2** — depending on the selected rotation direction) list on the Properties bar. The list of methods of defining the rotation angle contains the following options (they are described below):



▼ **At angle,**



▼ To vertex,

▼ To surface.

2.3.3.7.1. Rotation by the specified angle

Section rotation is done exactly by the angle specified in the **Angle** field.

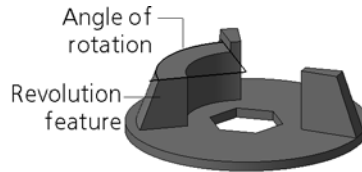


Fig. 2.3.27. Rotation by the specified angle

2.3.3.7.2. Rotation up to vertex

The section rotation angle is defined automatically by position of a vertex selected by user.

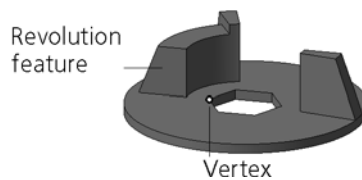


Fig. 2.3.28. Rotation up to vertex

- ▼ If a face, sketch, or contour built by sketch lines or face's edges, are used as a section, the rotation angle is calculated so that the element's endface (or its continuation) goes through the specified vertex.
- ▼ If an edge, 3D curve, or an arbitrary contour is used as a section, the rotation angle is defined as follows: In the plane, perpendicular to the axis, a circle is built, with its center on the axis, going through the specified vertex. A point nearest to the section is placed on this circle. The section's rotation angle is accepted as equal to the angle of the arc between the identified point and the specified vertex.



To specify a vertex defining rotation angle, activate **Vertex** switch on the Properties bar and select the desired point object in a graphic area (point can be also specified in the Design Tree). The name of the selected object is displayed in a field beside the switch.

Cut option allows to cut revolution feature by a plane going through the selected vertex and revolution axis. In this case, the element will have a flat endface irrespective of the section form.



Extension of complex shape faces is not always possible. Therefore, if the specified vertex is positioned in such a way that, provided the rotation axis is set, the element's endface does not go through the vertex, in some cases, construction will only be possible if the **Cut** option is enabled.

2.3.3.7.3. Revolution up to surface

The rotation angle is defined automatically by position of a face or a plane specified by User.

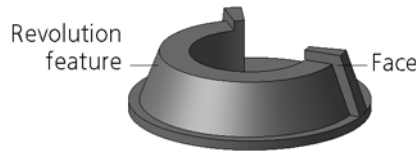


Fig. 2.3.29. Revolution up to surface



To specify the active surface, enable the **Face** switch and select the desired face or plane in the Design Tree or in the graphic area.

The element is rotated up to the selected surface. The endface of the feature has the surface form.

It is recommended to choose section, axis, and surface so that the surface fully intersects the rotation element. Otherwise, the construction is done only in case of selected surface can be extended. Extension of complex shape surfaces is not always possible.

2.3.4. Sweep and Cut-Sweep operations

2.3.4.1. Swept feature

A swept feature is created by sweeping section along the path, for example, as in Fig. 2.3.30.



Fig. 2.3.30. Section, guide and swept feature

Swept element may be an individual body or may be attached to a body or cut out of it.

Operation **Sweep** is used to create a new sweep body or to add a swept feature to an existing body (i.e. to add a material), operation **Cut-Sweep**— to cut a sweep feature from a body (to remove material).

2.3.4.2. Carrying out of Sweep Operation



1. Start an operation. To do this, complete the following steps:



▼ press the required button on the **Edit Part** or **Edit Model** panel.



▼ or call the required command from **Operations** menu.

Condition of availability of commands **Sweep** and **Cut-Sweep**: at least one sketch in the model. Additional availability condition of **Cut-Sweep** command: model including a body or a component.

After operation launch, tabs with operation parameters controls appear on the Properties bar.

2. Specify the operation parameters:

- ▼ **section** (see Section 2.3.4.3);
- ▼ **path** (see Section 2.3.4.4);
- ▼ **type of section movement** (see Section 2.3.4.5);
- ▼ **thin wall settings** (see Section 2.3.2.7),
- ▼ **result** (see Section 2.3.2.8).

All parameter values, once entered or modified, are immediately displayed on the screen as the ghost of the extruded feature. A ghost allows a visual control over parameter accuracy.



You can use handles to specify wall thickness (for more on handle see Section 2.1.4.2.2 on p. 138).



3. Click **Create Object** button to complete the operation.
4. If such operation results in a body comprising several parts, the Modify set of parts process will be started automatically after the end of the operation. Select the parts to be hold (see Section 2.3.11.1).

Operation result — new body, pasted or cut element — appears in the graphic area.

The operation's icon will appear in the Design Tree matching the icon on the command's button which is used to run the operation. Exception is an icon of the new swept body — it differs from an icon on the button.



— an icon of new swept body

2.3.4.3. Section of swept feature

A sketch is used as a section of swept feature. Requirements to the sketch are specified in Section 2.3.4.3.1.



To specify the swept feature section, activate the **Section** switch and select the required sketch in the Design Tree or in the graphic area.

Name of the selected sketch shall appear in the **Section** field on Properties toolbar.



Section can be specified before the operation launch.

2.3.4.3.1. Section sketch requirements

- ▼ Section sketch objects should comprise one contour.
- ▼ A contour can be closed or open.

2.3.4.4. Path of swept feature

The following may be used as a path of swept feature:

- ▼ 3D curve (or separate segment of multi-segment curve),
- ▼ a sketch line,
- ▼ an edge,
- ▼ Chain of objects listed above in any combination.



To set the path of section movement, activate \dots switch. Then, in the Design Tree or in the graphic area, specify the desired objects in the order they are connected.

The name of the selected object or the number of selected objects are shown in the **Path** field on the Property Bar.


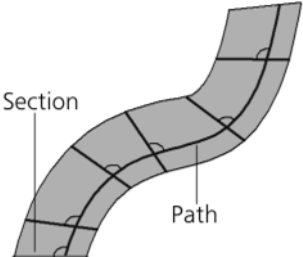
2.3.4.4.1. Path requirements

- ▼ A path can be closed or open.
- ▼ If path is open, its start point or end point should lie in the plane of the section sketch.
- ▼ If path is closed, it should intersect a plane of the section sketch
- ▼ A tangent to the path in its point common with the sketch plane should not be on the same plane.


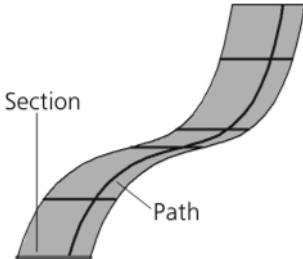

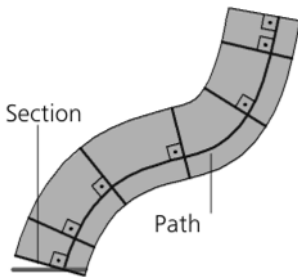
2.3.4.5. Type of Section Movement

While the sketch is being moved along the path, its orientation can be changed or kept the same. To specify the required type of section movement, activate the applicable switch in **Section Movement** group (see Table 2.3.7).

Tab. 2.3.7. Variants to change the section orientation during sweep

Switch in Section Movement group	Specifics of feature construction	Feature construction procedure
 Maintain Slope*	<p>When this option is selected, the section is swept at a constant angle to the path at any point along the feature equal to the angle between the original section sketch and the path at its start point.</p>	

Tab. 2.3.7. Variants to change the section orientation during sweep

	Switch in Section Movement group	Specifics of feature construction	Feature construction procedure
	Parallel to itself**	Section is moved in such a way that its plane remains parallel to the plane of the sketch containing this section at any point of the path.	
	Orthogonal to Path*	Section is moved in such a way that section plane remains perpendicular to the path at any point of the path.	

* If the plane of section sketch is perpendicular to the path in its initial point, then the **Maintain Slope Angle** and **Orthogonally to Path** options shall provide the same construction result.

** Carrying out section movement parallel to itself is not allowed if the path has points where a tangent to the path is parallel to the sketch section plane.

Fig. 2.3.31 shows construction of the swept feature under different section orientation relative to the path. Initial position of the sketch and the path are the same in all cases, but the results of construction are different.

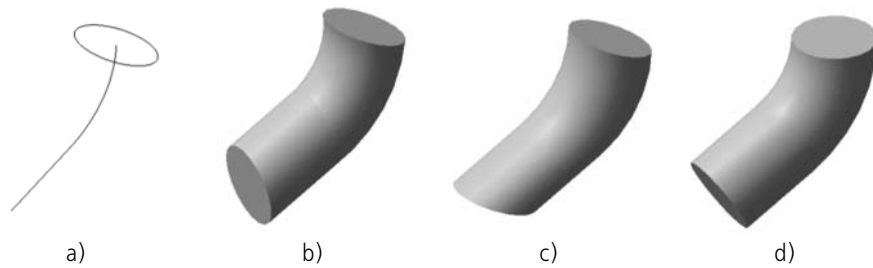


Fig. 2.3.31. Swept feature: a) sketches of section and the path, b) movement of the section with the slope angle maintained, c) movement of the section parallel to itself, d) movement of the section orthogonal to the path.

2.3.5. By Sections and Cut-Sections operations

2.3.5.1. Lofted Feature

Element by sections is created by connection of several sections of arbitrary shape and position, for example, as in Fig. 2.3.32. Extreme section can be a point.

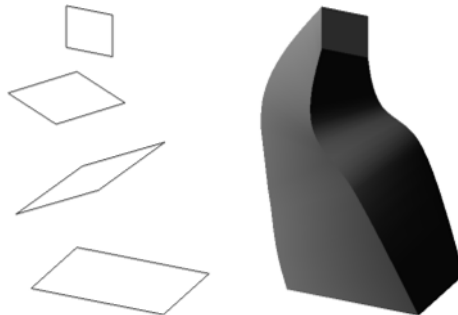


Fig. 2.3.32. Sections and element created using them

If necessary, you can use a centerline during element creation by sections (see Fig.2.3.33).

Lofted element may be an individual body or may be attached to a body or cut out of it.

Operation **Loft** serves to create a new lofted body or to add a lofted feature to an existing body (i.e. to add a material), operation **Cut-Loft**— to cut a lofted feature from a body (to remove material).

2.3.5.2. Execution of the operation



1. Start an operation. To do this, complete the following steps:

▼ click the required button on the **Edit Part** or **Edit Model** panel.



▼ or call the required command from **Operations** menu.

Condition of availability of commands **Loft** and **Loft-Cut**: at least two sketches in the model. Additional availability condition of **Cut-Loft** command: model including a body or a component.

After operation launch, tabs with operation parameters controls appear on the Properties bar.

2. Specify the operation parameters:

- ▼ **Section**, see Section 2.3.5.3;
- ▼ **Centerline** (see Section 2.3.5.4);
- ▼ **Element Construction Method near Extreme Sections**, see Section 2.3.5.5;
- ▼ **The Path of Sections Connection** (see Section 2.3.5.6);
- ▼ **Thin Wall Settings** (see Section 2.3.2.7),
- ▼ **Result** (see Section 2.3.2.8).

All parameter values, once entered or modified, are immediately displayed on the screen as the ghost of the extruded feature. A ghost allows a visual control over parameter accuracy.



A thin-wall loft feature can be created only if all its sketched sections contain contours. Such a feature cannot be created using point-containing sketches.



You can use handles to specify wall thickness (for more on handle see Section 2.1.4.2.2 on p. 138).



3. Click **Create Object** button to complete the operation.
4. If such operation results in a body comprising several parts, the Modify set of parts process will be started automatically after the end of the operation. Select the parts to be hold (see Section 2.3.11.1 on p. 226).

Operation result — new body, pasted or cut element — appears in the graphic area.

The operation's icon will appear in the Design Tree matching the icon on the command's button which is used to run the operation. Exception is an icon of new body by sections— it differs from an icon on the button.



— an icon of new lofted body

2.3.5.3. Sections of lofted feature

During lofted feature construction, sketches are used as a sections. Requirements to the sketches are specified in Section 2.3.5.3.1.



To specify the swept feature sections, activate the **Sections** switch and select the required sketches in the Design Tree or in the graphic area.

The list of sketches appears in the **Sections List** window in the order they were selected. The sections will be connected in this order when building the feature. To change the order of sections or delete some of them, use the buttons above the list.



While selecting sections in the feature window, indicate them in points (vertices) which shall be connected in sequence. In this case, when path is generated automatically (see Table 2.3.9), body of the required form will be built.



Sections can be specified before the operation launch.

2.3.5.3.1. Section sketch requirements

- ▼ Sketches can be positioned on arbitrarily oriented planes
- ▼ Every sketch objects should comprise one contour.
- ▼ Extremely (initial and final) located sketches may contain only one point (instead of a contour)
- ▼ The contours should all be either closed or open.

2.3.5.4. Centerline of lofted feature

If necessary, you can specify a centerline during lofted feature creation. Examples of elements built using the same sections with a center line and without it are shown in Fig.2.3.33.

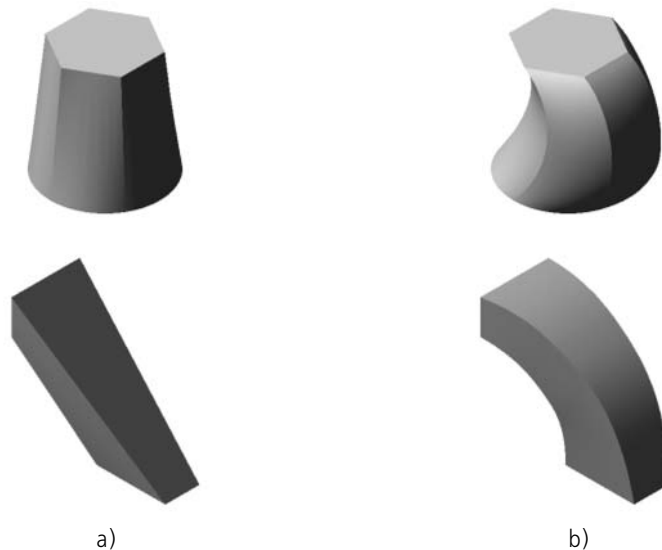


Fig. 2.3.33. Loft features: a) without centerline; b) with centerline

The following may be used as a centerline:

- ▼ a 3D curve,
- ▼ sketch,
- ▼ edge.

Centerline must satisfy the requirements presented in Section 2.3.5.4.1.



To specify the feature centerline, activate the **Centerline** switch and select the required object in the Design Tree or in the graphic area.

2.3.5.4.1. Centerline requirements

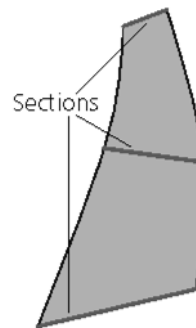
- ▼ Axis can be open or closed
- ▼ If the axis is open, its end points shall lie in planes of the first and the last section sketches
- ▼ If the section contours are open, the axis shall intersect the planes of section sketches in the points belonging to these contours. If section contours are closed, intersections within contours are acceptable.
- ▼ A tangent to the centerline in its point common with the sketch plane should not be on the same plane.
- ▼ If the centerline is a multi-segment one (for example, contour containing several curves), its segments must be connected tangentially or smoothly.

2.3.5.5. Method of Constructing a Feature near Extreme Sections

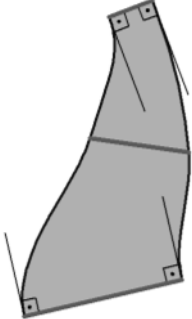
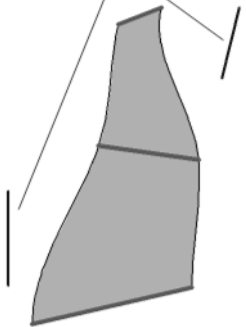
You can specify a direction of feature tangents going through the points of contours in its extreme sections. In other words, you can change the direction of feature “exit” from the first section and the direction of “entrance” into the last section. To do this, select the required option in **Initial Section** and **Final Section** list accordingly (see Table 2.3.8).

Tab. 2.3.8. Methods of constructing a feature by the initial and final section

Switch in Path group	Aspects of element construction	Feature construction procedure
By default	Selected vertices of the sections shall be connected by splines of third degree.	



Tab. 2.3.8. Methods of constructing a feature by the initial and final section

Switch in Path group	Aspects of element construction	Feature construction procedure
By Normal	The feature forms in such way that the plane tangent to its surface near the border (the first or the last one) section becomes perpendicular to its plane.	
By Object	The feature forms in such way that the plane tangent to its surface near the border sketch becomes parallel to the specified straight-line object or perpendicular to the specified planar object. To specify an object, activate the Vector of Initial (Final) Section switch and select the desired object in the Design Tree or in the graphic area.	Objects defining out of line sections: 


2.3.5.6. The Path of Sections Connection

The same sections can be connected by different methods. You can select a path generation method, as well as make it open or closed.

2.3.5.6.1. Method of Path Generation

Section points to be connected during element construction may be specified manually or defined automatically. To select the required variant, activate the applicable switch in **Path** group (see Table 2.3.9).

Tab. 2.3.9. Variants of path generation

Switch in Path group	Aspects of feature construction
 Automatic Path Generation	Section points to be connected during element construction are set automatically. This method is recommended if the topology of sections is the same (Fig.2.3.34).

Tab. 2.3.9. Variants of path generation

 Switch in **Path** group Aspects of feature construction



Path Generation by specified points Sketches are sequentially connected by points nearest to the points of their selection (Fig.2.3.35).



Fig. 2.3.34. Automatic connection of sections

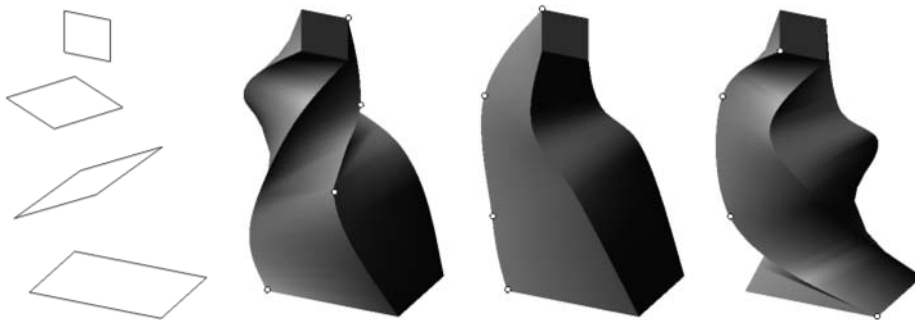


Fig. 2.3.35. features generated by connecting different points of similar sections (the connected points are highlighted)



If sketches are selected in the Construction Tree, the automatic path-generating algorithm is used.



If sections are not convex, select the path manually.

If topologies of sections differ significantly (i.e., one of them is a triangle, and the other one is a pentagon), the result of construction may not correspond to the expected result: the feature may be “coiled”, or additional edges may emerge (Fig.2.3.36).

To fix the above mentioned error, edit the sections as follows:

1. Identify the section whose contour contains the largest number of segments, i.e. geometrical objects (straight line segments, arcs, splines). Determine this number.
2. Divide geometrical objects in other sections to satisfy two conditions:

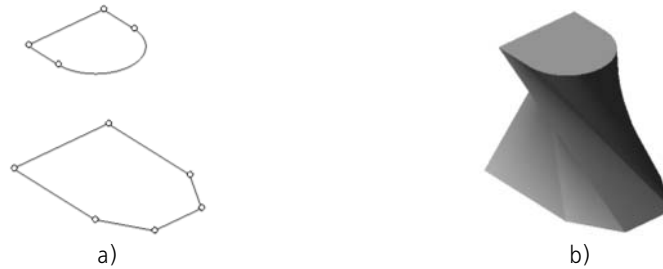


Fig. 2.3.36. Automatic connection of sections with different topology

- ▼ Each section shall have the same number of segments, i.e. geometrical objects. It shall be equal to the number defined in the item1.
- ▼ The splitting points shall lie on the presumed edges of the loft.

For methods of dividing geometric objects, see Section 3.4.6 on p. 1136.

The resulting feature based on modified sections is shown in Fig.2.3.37.



Fig. 2.3.37. The arc in the upper section is divided into three mating parts.

As a result, the contour in the section has not changed, while construction result has become predictable

2.3.5.6.2. Closed or Open Path



To select an option for path construction, activate **Open** or **Closed** switch in **Mode** group.

Upon constructing a closed path, sections specified as the first and the last shall become connected.

If centerline (see Section 2.3.5.4 on p. 179) is not in use, **Mode** group is available if more than two sections are selected.

If a centerline is selected in addition to sections, then the **Mode** group is inaccessible. At the same time, the active switch is left set to the axis configuration.

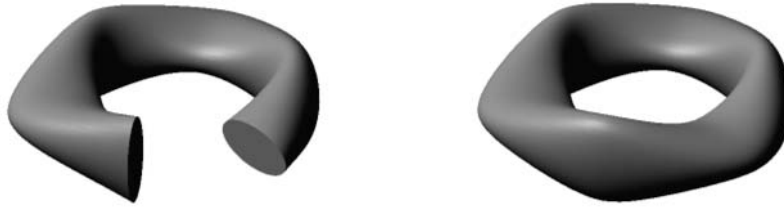


Fig. 2.3.38. Open or closed features constructed by the same sections



If a feature is closed, the method to construct it incorporating all sections shall be **By Default** (see Section 2.3.5.5).

2.3.6. Thickening a surface or body face

You can thicken body faces or a surface, i.e. add a material layer to these faces. Faces selected for thickening may belong to either objects of the model itself, or those of its components.



For adding a material layer, use the **Thicken** command.

The command is accessible if there is at least one surface, body or component.

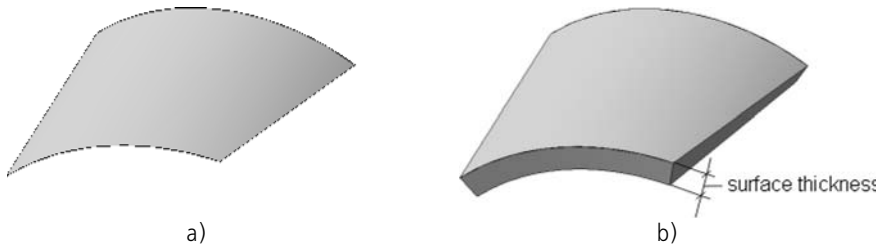


Fig. 2.3.39. Thickening a surface face:
a) the initial face, b) adding a material layer in one direction

The following objects may be used as source ones:

- ▼ face of the body or surface,
- ▼ a connected set of faces of one body or surface.

If you want to add a material layer to all faces or to the entire surface, this body or surface must be selected in the Design Tree.

Select the source object.



The source objects for adding material can be also selected before calling the command.



The second selection of a face cancels it's selection.

If you need to change all the objects to be thickened, click the **Reselect** button on the Special Control Panel and then reselect the objects.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.

After the source objects are specified, there will be displayed the ghost of the added layer and the thickness control handles.

Specify thickness of the added layer. It is made in a similar way as when setting up thin wall parameters (see Section 2.3.2.7 on p. 161), with the only exception that: you can not cancel the construction of thin wall by clicking **No** option.

If necessary, customize the operation application area using the controls of the **Operation Result** tab on the Property Bar (see Section 2.3.10.3.1 on p. 220).

The construction is impossible in the following cases:

- ▼ the added material layer is a body with damaged topology, i.e. with breaks or the surface parts degenerated into a point or line;
- ▼ the surface of the added layer intersects itself;
- ▼ when several faces are specified, the intersection of their equidistant surfaces can't be defined uniquely.



Click the **Create Object** button to fix the added material.



The created feature will appear in the model window and the applicable icon will appear in the Tree.

The operation results in the material layer restricted with the initial surface and equidistant to it or with two equidistant surfaces.



It is impossible to thicken faces of a body which consists of several parts. You must first make the body integral by editing the existing elements or by creating new elements to connect the parts.

2.3.7. Boolean operation applied to bodies

Serves to perform a Boolean operation on two bodies present in the current model. Bodies involved in the operation may belong to either the model itself, or its components. The result of the Boolean operation is a new body that belongs to the current model. This body may be used in any future operations, including the Boolean ones.

The command is available if there is more than one body in the part.



To call the command, click the **Boolean Operation** button on toolbar for editing the model or select its name from **Operations** menu.

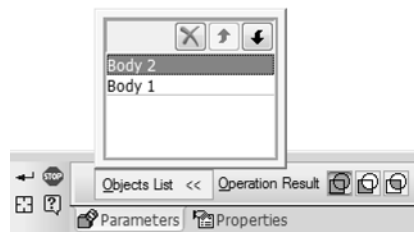


Fig. 2.3.40. Properties toolbar during Boolean operation execution

Select the first and the second body involved in the operation.

Selected bodies are highlighted in the Model window. Their respective icons will be highlighted in the Design Tree. The names of the objects belonging to bodies are displayed in the **List of Objects** on the **Settings** tab of the Properties toolbar (Fig. 2.3.40).



If you need to cancel the selection, click the **Reselect** button on Special control panel. Highlighting of all selected bodies will be cancelled, and you will be able to select the new ones.



If only the second body was selected incorrectly, simply select the desired one without cancelling the previous selection.



Define the Boolean operation type using the **Operation Result** switch group.

Activate the **Union** switch to merge the selected bodies.



To delete the first body volume portion which is occupied by the second body, activate the **Subtraction** switch. The body appearing first in the list is considered to be first, the body positioned second is considered to be second.



To change the sequence of bodies, use **Move down** and **Move up** buttons on the **List of bodies** panel.



To delete all volume of both bodies except for their common volume, activate the **Intersection** switch.



Bodies can be merged if they intersect or have a common surface, whereas subtracting and intersecting are possible only if bodies intersect.

The object name, i.e., the name displayed in the Design Tree, can be defined on the **Properties** tab of Properties panel.



When you have set the operation parameters, click the **Create Object** button on the Special Control panel.



The Design Tree will show an icon of a Boolean operation, and in the window of part the body being the operation result will appear.



A Boolean operation of subtraction or intersection of bodies can result in a body made up of multiple pieces (see Section 2.3.11 on p. 224).

2.3.7.1. Display of the Boolean operation on the Design Tree

Fig. 2.3.41 presents examples of displaying Boolean operations on the Design Tree for bodies on various levels of the model's structure.

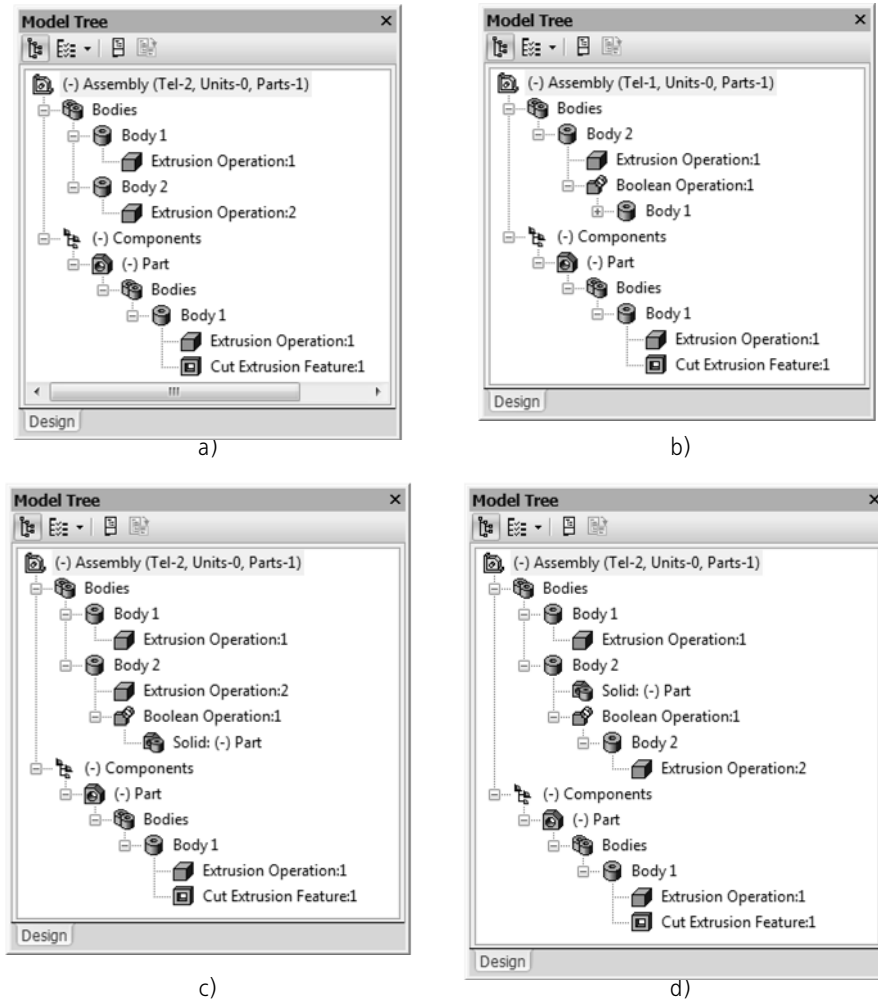


Fig. 2.3.41. Displaying Boolean operations on the Design Tree: a) initial state of the model, b) Boolean operation on bodies belonging to the model, c) operation on a body belonging to the model (specified as the first) and a body belonging to the component (specified as the second), d) the same, but the order of specifying the bodies is reverse

2.3.8. Body editing operations

Functionally, KOMPAS-3D has taken into account the specific engineering drafting work techniques. The system is targeted at producing models of certain parts containing typical structural features, which simplifies performing certain specific operations. These operations include, for example, creating a chamfer, a rib, etc.

To simplify the process of defining parameters for those features, dedicated commands are provided for creating them. Thus, to create a chamfer, you do not have to draw a sketch, sweep it along an edge and subtract the resulting volume from the main body. It is sufficient to select

chamfer edges and then enter its parameters, i.e. the lengths of the sides or a side length and an angle.

Commands to create structural features are placed in **Operations** menu.



To specify numerical parameters of additional structural features, you can use handles (see Section 2.1.4.2.2 on p. 138).

Editing operations apply both to bodies built within the model itself, and those built within components.

2.3.8.1. Chamfer



To create a chamfer on the part edges, apply **Chamfer** command.



The command is not performed on edges created by smoothly adjoining faces.



Select a method to construct a chamfer— **By Side and Angle** or **By Two Sides** by activating the corresponding switch in **Method of Construction** group.



- ▼ If a chamfer is created by a side length and an angle, enter the chamfer side length in the **Length 1** field, and in the **Angle** field the angle between the side and the chamfer surface. The calculated value of the chamfer's second side shall appear in the **Length 2** reference field.

- ▼ If a chamfer is created by two side lengths, enter their values in the **Length 1** and **Length 2** fields. The calculated value of the chamfer's angle shall appear in the **Angle** reference field.

Select edges to create a chamfer in the Part window. If you need to create chamfers on all edges of a face, select that face.

Upon selecting the first edge, a ghost chamfer is displayed in the Part window together with an arrow pointing along one of the faces. The arrow indicates direction in which the chamfer side with **Length 1** length shall be plotted. The chamfer angle shall be plotted relative to this direction.



If changing the direction in which the first side is to be plotted is required, activate the applicable switch (**First Direction** or **Second Direction**) in group **Direction of the first side**. The direction of the ghost arrow (hence, the direction of the chamfer's first side as well) shall change.



If sides of a chamfer are equal, then the result of it's construction shall not depend on the first side direction.



Do not create a chamfer for each edge individually. Specify as many edges with the same face parameters as possible while creating a chamfer.



You may find it difficult to follow this advice if a chamfer is not equilateral. If edges selected to construct such a chamfer do not belong to one face, then selecting a uniform chamfer direction for all edges may result in a wrong construction. In this case, we recommend selecting edges belonging to one face to construct a chamfer, and creating individual chamfers for each of the edges.

If several edges used to construct a chamfer are merged smoothly (have a tangent where they connect), select one of them and enable the **Extend Along Tangent Edges** option. In this case, the system will automatically determine other edges on which to extend the chamfer.



Once operation execution is complete, a chamfer shall appear on the edges of the part, and a chamfer icon— in Model Tree.



Create chamfers and fillets, wherever possible, at the end of part construction process, rather than immediately upon emergence of shape-generating features requiring creation of chamfers and fillets. In this case, the calculations during execution of shape-generating feature operations shall be performed faster.

2.3.8.2. Fillet



To round an edge (or several edges), apply the **Fillet** command.



The command is not performed on edges created by smoothly adjoining faces.



Select **Type of Fillet** — with constant or variable radius. By default, **Constant Radius** switch is enabled. If you need to create a variable-radius fillet, activate the **Variable Radius** switch.



Select the edges for filleting. If you need to fillet all edges limiting any face, select this face.



Fig. 2.3.42. Fillet: a) to construct fillet, the upper face is shown, option is enabled; b) result of fillet construction



Do not fillet each edge individually. If possible, select the most number of edges to fillet using the same radius. In this case, calculations will be run faster upon model rebuilding.

To create a fillet with constant radius, select its creation method and set its parameters (see Section 2.3.8.2.1 on p. 190).

To create a fillet with variable radius, set additional parameters on **Variable Radius** tab on Properties toolbar (see Section 2.3.8.2.2 on p. 192).

Activate the **Parameters** tab of the Property Bar and select corner capping and corner treatment methods (see Section 2.3.8.2.3 on p. 193).

If several edges to be filleted merge smoothly (are tangent at the connection), select one of them and enable the **Extend Along Tangent Edges** option on the Options tab of the Properties toolbar. In this case, the system will automatically determine other edges to which to extend the fillet.



Fig. 2.3.43. Fillet: a) to construct fillet, one edge is shown and **Continue by Tangent** option is enabled; b) result of fillet construction

The **Maintain Edge** option allows to create a fillet while maintaining the shape of neighboring face edges (see Section 2.3.8.2.4 on p. 194). The option is unavailable when creating the fillet with variable radius.

If you need to stop filleting at a certain point on the edge, activate the **Stop Filleting** tab and set parameters for the stopping point (see Section 2.3.8.2.5 on p. 195).


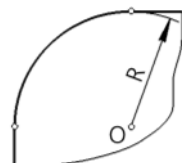


As soon as completion of operation is confirmed, part edges will be filleted and a fillet icon will appear in Design Tree.



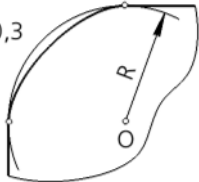
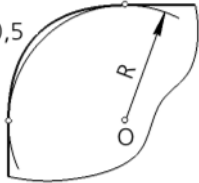
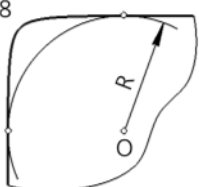
2.3.8.2.1. Fillet with Constant Radius

A fillet with constant radius can be created by four methods (Table 2.3.10). To select the right method, activate the corresponding switch in the **Method** group on the Property Bar.


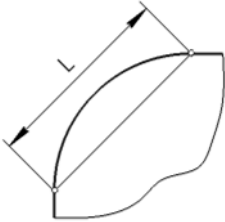
Tab. 2.3.10. Methods of creation of fillet with constant radius

Method	Construction result
 <p>With Circle Arc The fillet profile is a circle arc. Enter the required fillet radius in the Radius field or define it using the counter.</p>	

Tab. 2.3.10. Methods of creation of fillet with constant radius

Method	Construction result
	<p>With Elliptical Arc The fillet profile is an elliptical arc. Enter the required values of ellipse semiaxes (L1,L2) in the fields Semiaxis 1 and Semiaxis 2 or use the spinner to select them.</p>
	<p>With Factor The fillet profile is a conic curve. Enter the required values of fillet radius and of conic fillet curve factor K in the fields Radius and Factor or use the spinner to select them. The K value can vary from 0 to 1 ($0 < K < 1$). If the K factor value is more than 0 but less than 0.5 ($0 < K < 0,5$), the fillet profile is an elliptical arc.</p>
	<p>K=0,3</p> 
	<p>If the K factor value is equal to 0.5, the fillet profile is a parabola.</p>
	<p>K=0,5</p> 
	<p>If the K factor value is more than 0.5 but less than 1 ($0,5 < K < 1$), the fillet profile is a hyperbola.</p>
	<p>K=0,8</p> 

Tab. 2.3.10. Methods of creation of fillet with constant radius

Method	Construction result
 <p>With Constant Chord The fillet profile is a circle arc. Enter the required chord value in the Chord field or use the spinner to select it.</p>	

2.3.8.2.2. Fillet with a Variable Radius

To create a fillet with a variable radius, you need to define points on the edges being filleted and the filleting radii in these points.

A panel containing a table of fillet parameters of the same name is located on the **Variable Radius** tab. It includes point IDs, distances from them to the starting points of the respective edges and fillet radii values in these points (Fig.2.3.44, b). As long as the points to construct a fillet are not specified, the table of parameters is empty.

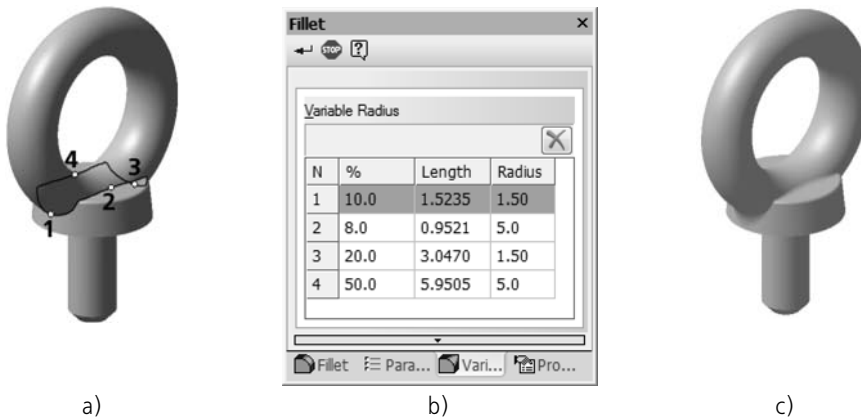


Fig. 2.3.44. Variable-radius fillet parameters: a) selection of points, b) fillet parameterization, c) command execution result

Select the desired points in the Model window. Selected points shall be marked with crosses and numbered in order of their selection (Fig.2.3.44, a).


In **Variable Radius** table, specify values of fillet radii in selected points.

If necessary, specify the values in **%** or **Length** column. Remember that changing of values in these columns moves the applicable point in the model window.



To change any value, click once or double-click in the required cell. Upon a single click, you can enter values in the cell by typing from the keyboard, upon a double click — by typing from keyboard or selecting by the stepper button.



You can delete the table row and, consequently, a point in the part preview window. To do this, click the  button placed over the list.

Setting of parameters of a fillet with a variable radius has the following specific features.

- ▼ By default, the fillet radius at the boundary edge points is equal to the default radius defined in the **Radius** field on the **Fillet** tab of the Property Bar. Boundary points of an edge are its start and end points. These correspond to 0% and 100% of the edge length. If a fillet radius at the boundary point has to differ from the default, select the point explicitly in the Model window and define the desired radius value in the Fillet parameters table.



If an edge is closed, then its start and end points coincide, and you can select just one of them when setting a fillet.



If two edges are selected to construct a fillet, where the end point of one of them coincides with the start point of another one, then you can select just one of them when setting the fillet.


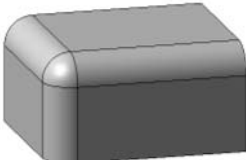
- ▼ The zero radius can be defined only at a fillet boundary points. If it is difficult to precisely select a boundary point of an edge in the Model window, then you can make a rough selection first, and then enter the desired value— 0% or 100%— in the respective entry of the Parameters table. Note that you cannot set a zero radius for a non-boundary point, nor can you change the distance from the vertex for a point with the zero radius (to do this, change the radius first).
- ▼ When rounding edges without specified points, the default value is used. For example, if the **Extend Along Tangent Edges** option on the **Parameters** tab of the Property Bar was enabled when creating a fillet with variable radius, such tangent edges will be automatically found by the system. Since it is impossible to specify points on them, those are filleted with the default radius.

2.3.8.2.3. Corner Capping and Treatment


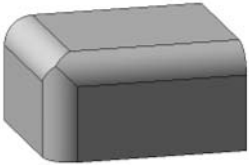
Capping and **Treatment** are fillet face joining methods.

The methods are selected on the **Parameters** tab of the Property Bar using the group of switches **Capping** and **Treatment**. Construction examples are presented in Tables 2.3.11 and 2.3.12.


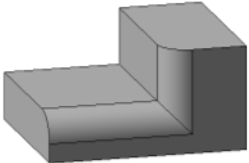

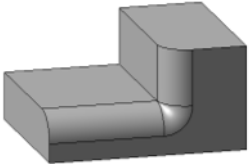
Tab. 2.3.11. Corner Capping Methods

Method	Construction result
 Smooth angles	

Tab. 2.3.11. Corner Capping Methods

Method	Construction result
 Don't smooth angles	

Tab. 2.3.12. Corner Treatment Methods

Method	Construction result
 Passing without smooth joining	
 Passing with smooth joining	



The **Treatment** group is available if an edge is filleted by the method **With Circle Arc** (see Table 2.3.10 on p. 190).

2.3.8.2.4. Fillet with a maintained edge or surface

If the fillet surface intersects with adjacent faces, there are two ways of performing the operation:

- ▼ with a maintained edge;
- ▼ with a maintained surface.

To select the first variant, enable the **Maintain Edge** option on the **Parameters** tab of the Property Bar. In this case, the shape of neighboring face edges will be saved (Fig.2.3.45, b).

If this option is off, fillet shall be created with surface saving. The shape of neighboring face edges may change (Fig.2.3.45, c).

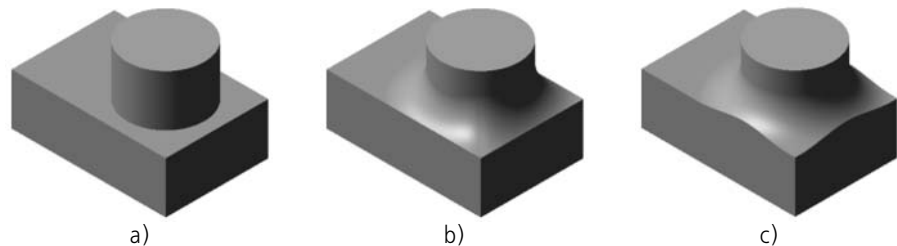


Fig. 2.3.45. Fillet: a) initial features, b) fillet with saving the edge, c) fillet without saving the edge

Sometimes the relative disposition of rounding surfaces and specified radius of the fillet do not allow saving the edge. If, in such a case, the **Save Edge** option is enabled, then any attempt to create a fillet shall produce a system error message.

To avoid such an error, enable the **Auto Determination** option. In this case, the system will create a “composite” fillet. Wherever possible, edges of neighboring faces shall be saved. Where saving of edges is impossible, surfaces of neighboring faces shall be saved.

If auto determination is on, the **Save Edge** option is unavailable and its state is ignored when constructing a fillet.

2.3.8.2.5. Stopping filleting

You can stop filleting if it involves a single edge (or a chain of edges). You can specify one or two stopping points on an open edge (chain), it is mandatory to specify two points for a closed edge.

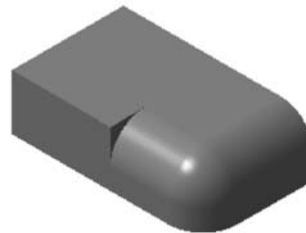


Fig. 2.3.46. Stopping filleting



To stop filleting at a certain point on the edge activate the **Stop Filleting** tab on the Properties panel and enable the **Stopping** option.

The tab has the **Stopping points** panel. The panel contains a table for setting stopping point parameters: point numbers, distances to them from the starting point of the edge (or chain) being filleted, information about associative links with the object.

Set a fillet stopping point(s). To do so, you can use the following methods:

- ▼ specify an arbitrary point on the edge,
- ▼ specify a vertex that belongs to the chain of edges being filleted,
- ▼ specify an object for associative connection:

- ▼ point object belonging to the edge being filleted,
- ▼ 2D object intersecting the edge being filleted (the stopping point in this case is the point of intersection of the edge with the said object).

The set stopping point is marked in the model screen by a number and an arrow. The arrow shows the direction of filleting: that part of the edge where the arrow points at will remain unchanged, while the rest of it will be filleted.


In the table on the Properties bar the stopping points are numbered in the order they are specified. If a point is specified arbitrarily, the values in the **%** or **Length** column can be edited. As you change the values, the point will move on the model screen.

If there is an association link between the stopping point and the object, the **Object link** column will have a check mark, and it is impossible to change values in the **%** and **Length** columns. When you highlight the row in the table that corresponds to the point linked with the object, a switch and a field named **Object** appear on the Properties bar. The switch will have a check mark, and the field—name of the object linked with the point. To cancel the link, you need to either click in the cell of the **Object link** column, or the **Object** switch. Note that to restore the link you need to select the object again.



You can change the fillet truncation direction to the opposite by clicking the **Change direction** button above the list.

If the 2D object using which the stopping point is determined intersects the edge being filleted in several points, the **Next** and **Previous** buttons become available. They are used to select the appropriate point.

You can delete the table row and, consequently, a point in the part preview window. To do this, click the  button placed over the list.

By default, the filleting is truncated in the stopping point by an intersecting plane perpendicular to the edge (Fig. 2.3.46). If the stopping point is set as the point of intersection of the edge with a 2D object, the fillet can be trimmed by that object's plane. To do so, select the row that corresponds to that point and enable the **Object trimming** option.

2.3.8.3. Body transformation into shell

At any stage of work, a body (or several bodies) can be transformed into a thin wall shell.

When creating a shell, the entire part body is excluded from calculations while a layer of material is added on its sides creating a shell.

To create a shell, you need to exclude one or several faces from adding material. These faces will become holes (or a hole) in the resulting shell.



When working with a body made up of multiple pieces, you can exclude faces of some of them. These pieces are transformed into shells with holes while the rest of them are transformed into closed shells.



To create a thin wall shell, apply the  command.



Select direction of material adding— **Outward** or **Inward** relative to the part surface. To do this, activate the applicable switch in **Thin Wall Type** group.



If material is added outwards, input shell thickness into the **Wall Thickness 1** field; if inwards— into the **Wall Thickness 2** field.

Select body faces that should not be involved in shell creation. The number of these faces is displayed in the **Number of Removed Faces** reference field.

To exclude some face from being selected, click it in the part window again. The face will be deselected and not removed when creating the shell.



To deselect all faces, click the **Reselect** button on the Special Control panel.

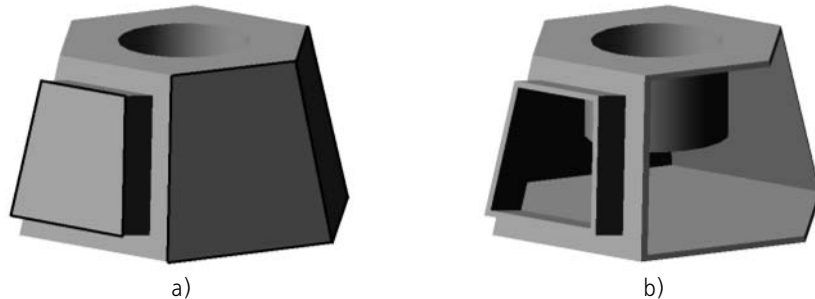


Fig. 2.3.47. Creation of a thin wall shell: a) initial part, removed faces are selected, b) result of thin wall shell creation



Once execution of the operation is confirmed, the Model Window will display image of a thin wall shell and the Tree will display its respective icon.

Working with the resulting shell will be the same, i.e., adding and subtracting bodies, creating chamfers, fillets and holes.

2.3.8.4. Rib

Before constructing a rib in the model, you need to create a sketch defining the shape of the rib. The rib is constructed from the sketch's line to the body. As a result, a thin wall is formed limited by the sketch's line on the one side, and by the body's faces on other sides.

2.3.8.4.1. Rib Sketch Requirements

- ▼ Sketch objects should comprise one contour.
- ▼ The contour must be open.
- ▼ Tangents to a contour in its end points must intersect the body.
- ▼ The contour does not have to reach the body. The system shall continue the contour up to the intersection with the nearest face. Curvilinear contours are extended tangent to their end points.

2.3.8.4.2. Generating a Rib



To create a rib, call the **Rib** command.



The **Rib** command is accessible if one sketch (rib sketch) is selected.

If a part consists of multiple bodies, then, to create a rib, you need to specify the one to which the created edge shall be attached. You can attach edges both to the bodies built within the model itself and ones built in the components. Note that faces limiting an edge should belong to the same body.

To select a body, select any face, vertex or edge of that body in the model window or select in the Design Tree an object belonging to that body of any of the following types:

- ▼ First shape-generating feature
- ▼ Shape-generating boss feature
- ▼ A boss sheet body
- ▼ glued mirror copy of a body
- ▼ Boolean operation.

The selected object will be highlighted.



It is most convenient to select a face (or one of the faces) to which the rib will be connected directly in the model window.



If you accidentally made a mistake when selecting body points, click the **Reselect** button on Special control panel. Highlighting will be cleared off the previously selected object and a new body may be selected.

Using control features of Properties toolbar, set the rib parameters:

- ▼ position— on the **Parameters** tab (see Section 2.3.8.4.3),
- ▼ face draft—on the **Parameters** tab (see Section 2.3.8.4.4),
- ▼ thickness— on **Thickness** tab; it is specified in a similar way as when setting up thin wall parameters (see Section 2.3.2.7 on p. 161), with the only exception that: you can not cancel the construction of thin wall by clicking **No** option.


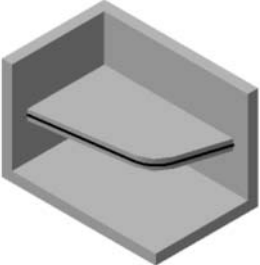

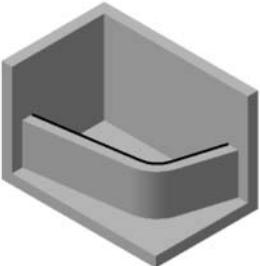


Once execution of operation is confirmed, a rib will appear in part preview window, and its respective icon will appear in the Tree.

2.3.8.4.3. Position of Rib

Rib may be perpendicular or parallel to the sketch plane. To select a construction option, activate the applicable switch in **Position** group (see Table 2.3.13).

Tab. 2.3.13. Possible rib positions

	Value of Position option	Rib position and shape	Construction result
	In the Plane of sketch	Rib is parallel to the sketch plane. A sketch constrains the rib along perimeter.	
	Orthogonal to Sketch	Rib is perpendicular to the edge sketch plane. Rib surface is created by sketch extrusion.	



If the part body is on one side from the sketch, the system shall automatically determine the direction of the edge extrusion (to the part side). This is the **Forward Direction**. To change direction of rib construction, activate the **Reverse Direction** switch in **Direction** group. Direction of rib construction is shown on a ghost in the model window as an arrow.

2.3.8.4.4. Draft of Rib Faces

If the feature faces are to have a draft, enter its value into **Draft Angle** field. Side faces of the rib shall be sloped outwards under specified angle.

If the rib sketch consists of several segments (straight lines), you can select a segment defining the draft direction. To do this, use the **Next** button. Upon clicking it, one of the sketch segments is highlighted. Click the button until the required segment is highlighted. The narrow side face of the rib corresponding to that segment will be accepted as the draft base. The side faces of the rib will be drafted with respect to it by the specified angle (Fig.2.3.48).



Selecting a segment to provide the draft direction makes sense only if rib construction is done in the sketch plane (when **In the sketch plane** switch is on).

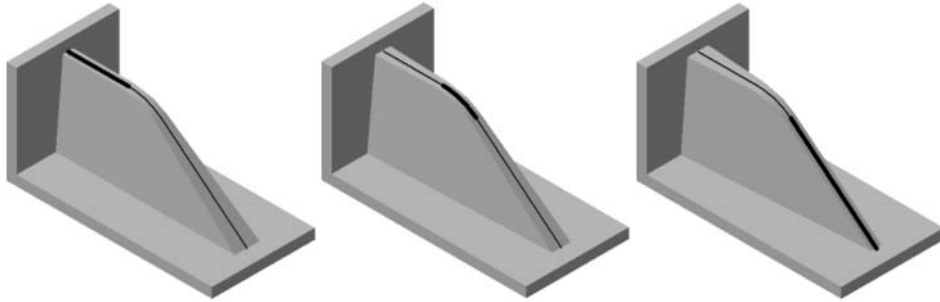


Fig. 2.3.48. Rib with a slope. Rib sketch consists of three segments. Segment defining draft direction is highlighted.

2.3.8.5. Face Draft



To apply a slope to flat faces perpendicular to the base or cylindrical faces whose constituents are perpendicular to the base, call the **Draft** command.

To construct a draft, you should specify its base and sloping faces.

Draft base is a flat face of the part whose shape, dimensions and the inclination angle do not change after performing the **Draft** command.

Sloping faces are faces whose inclination angle with respect to the base will change as a result of performing the **Draft** command.



To select a base, activate the **Base** switch and select the required face in the part preview window.



To select faces to draft, activate the **Faces** switch and select the desired faces in the Part window.

There is always only one base. There can be several sloped faces.



Sloping faces should merge with the base. They may be not merged with each other.



Select direction of sloping faces — **Inward** or **Outward** — by activating the applicable switch in the **Draft** group. If faces slope inwards, the feature section is decreasing, if outwards— increasing.



Sometimes, different faces can be selected as the base. In this case, the result of the command will depend on the relative position of the base and the sloping face (Fig.2.3.49).

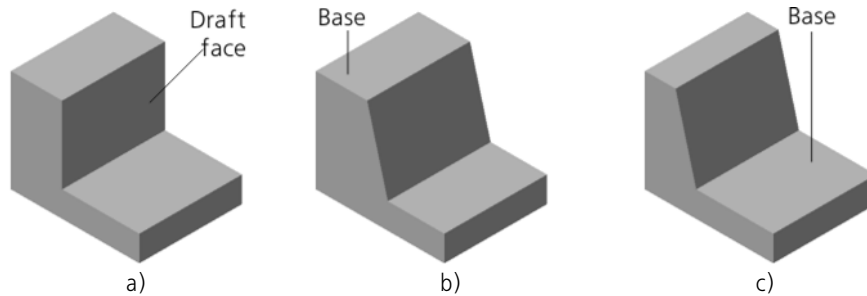


Fig. 2.3.49. Draft: a) initial part and sloping face, b) and c) result of creating a draft depending on base

Enter the inclination angle value in the **Angle** field.



If the **Draft** command is applied to a face already drafted with respect to the base by some angle, that angle will be discarded, i.e., the result will be the same as if the command was applied to a face perpendicular to the base.



Once execution of operation is confirmed, the selected faces shall obtain a slope, and a matching icon shall appear in Design Tree.



Do not slope faces individually. If possible, select the most number of faces to be drafted by the same angle with respect to the same base using the **Draft** command. In this case, calculations will be run faster upon model rebuilding.



The command is not performed if the system finds that the faces rebuilt according to the specified slope parameters do not form a body.



If you need to fillet one or several edges bounding the sloped face, do this after applying the slope on the face.

Using the **Draft** command is most efficient at final stages of designing diecast parts, when certain faces need to be drafted in order to ease casting separation from the form.

Generally speaking, sloping to side faces of extrusion feature can be done by editing parameters of this feature. However, this method differs significantly from execution of sloping operation:

- ▼ The **Draft** command lets bending separate faces, while all side faces are bending at an extrusion with a draft.
- ▼ The **Draft** command executing does not change position of features subjected to sloping faces, and editing the extrusion feature with sloping faces “bends” the objects subjected to these faces.

2.3.8.6. Trimming a part of the model

At any stage of part modelling, you can delete a component of its body (bodies). The border of section can be a surface of a free form or a cylindrical surface created by extrusion of an arbitrary sketch.

Note that the plane or surface used for cutting off can belong either to the model itself, or any of its components, while a sketch must belong directly to the model.



Generally speaking, these actions can be performed using the **Cut off by Extrusion** command. However, this will require setting a greater number of parameters and, in some cases, a more complex sketch than it is necessary for cutting off part of the model.

To cut a model component off by an intersecting plane or an extrusion surface, special commands are provided. They are located in the **Operations— Section** menu. Shortcuts are placed in one group on **Edit Part** toolbar.



Position and form of a cutting surface may be such that the section operation may result in a body made up of multiple pieces (see Section 2.3.11 on p. 224).

2.3.8.6.1. Section by Surface



To delete a model component located on one side of the intersecting surface, apply the **Section by Surface** command.

The following objects can be used as a base point:

- ▼ Base and auxiliary surfaces
- ▼ Surfaces imported and created in a part
- ▼ faces.



Flat surface can be lengthened in any direction, which is why it is not necessary to place it to intercept the whole item. Lengthening of a non-flat surface is not allowed, therefore the correct cut of an item part is only possible when it intercepts the whole item body.

If, before calling the command, a surface intersecting with the model was selected, then the surface name will appear in the **Section Surface** field on the Properties toolbar.

If no surface was selected before calling the command, select one.



You can delete a component of a part on any side from the surface of section. To select direction, activate the applicable switch in the **Direction of Cutting**— or group. Direction of cutting is shown on a ghost in the part preview window as an arrow.



Fig. 2.3.50. Cutting a component of the part by plane:
 a) initial state of the part (cutting plane is denoted relatively),
 b) result of cutting operation

If you are working with a multibody part, setting of an operation application area may be necessary as well as parameter setup. For this, the **Operation Result** tab of the Properties panel is used. Detailed information on operation application area and methods of its specification is provided in Section 2.3.10.3 on p. 219.



Once execution of operation is confirmed, the model shall be cut off, and a corresponding icon shall appear in the Design Tree.

2.3.8.6.2. Section by Sketch



To delete a part component lying to one side of an extrusion surface intersecting this part, use the **Section by Sketch** command.



Cutting surface is created by moving the sketch in the direction perpendicular to its plane.

Before calling the command, select the sketch that contains the profile of the ruled surface.

Section Surface Sketch Requirements

- ▼ Sketch objects should comprise one contour.
- ▼ The contour must be open.
- ▼ The contour should intersect the part projection on the sketch plane.

Section



Call the **Section by Sketch** command.

The name of the sketch that will be used for cutting off appears in the **Section Profile** field on the Properties toolbar.



You can delete a component of a part lying to any side from the extrusion surface. To select direction, activate the applicable switch in the **Direction of Cutting**— or — group. Direction of cutting is shown on a ghost in the part preview window as an arrow.





Fig. 2.3.51. Cutting a part component by sketch:
 a) initial state of the part, the sketch of the surface to be cut off is shown,
 b) result of cutting operation

If you are working with a multibody part, setting of an operation application area may be necessary as well as parameter setup. For this, the **Operation Result** tab of the Properties panel is used. Detailed information on operation application area and methods of its specification is provided in Section 2.3.10.3 on p. 219.



Once execution of operation is confirmed, the model shall be cut off, and a corresponding icon shall appear in the Design Tree.

2.3.8.7. Scaling of bodies and surfaces

Scaling allows changing body or surface dimensions in three directions according to the set factor. The result body or surface can be used with no restrictions in any future operations, including the scaling operation.

The **Scaling** command is available if there is at least one body or surface in the model. Scaling operation is also applicable to component bodies and surfaces.



To perform a scaling operation, click the **Scaling** button on the Edit Model toolbar or select its name from the **Operations** menu.

Select the body or surface to be scaled in the Design Tree or Window.



The body icons and names are shown on the Tree if the model structure is enabled (see Section 2.1.2.2 on p. 106).

If a model contains one body or surface, this body or surface is selected automatically.

The selected object is highlighted in the Design Tree and in the Model window. An outline box is also displayed in the Model window; you can visually estimate scaling by its further change.

The **Body or Surface** field on the Property bar displays the object name.



You can change the dimensions of one body or one surface in a single **Scaling** command call.

Input or select the scaling factor in the counter.



The scale factor can be also set using a handle (see Section 2.1.4.2.2 on p. 138).

In the Design Tree or in the Model window, select the scaling center: a vertex or a point in the space specified as a fixed one. Upon completing this command, the distance between this point and the center of gravity of the scalable object will change proportionally to the scale factor and in the direction of the line segment connecting these points.

By default, the origin of the current coordinate system is the centerpoint.

The point is highlighted in the model and its name is displayed in the **Center Mark** field.

To change the scaling centerpoint, select a new point in the Model window.



You can create a point by clicking the **Point Drawing** button on the Special Control Panel (for details see Section 2.5.2.11 on p. 346).

During the process of entering the scaling parameters, the outline box shall change its dimensions and position relative to the scaling centerpoint and proportionate to the specified factor.



To select other objects for this operation, click button **Select other** on the Advanced Control Panel. The current object selections will be cancelled allowing you to specify new objects.

Object name, color and visual properties of its surface can be specified on the **Properties** tab of the Property Bar.



When you have set the operation parameters, click **Create Object** button on the Special Control panel.



The scaled body or surface shall appear in the Model window and the respective icon shall appear in the Design Tree.



Please note that only bodies themselves or surfaces can be subjected to scaling. Their source and derivative objects - sketches, thread notations, symbols, etc.— remain unchanged.



While further working with the model created using the **Scaling** command, the following data shall be accounted.

- ▼ A scaled **sheet body** loses its respective properties specific for this body type.
- ▼ When creating a **object arrays**, previous scaling operations on the object will be ignored. Sizes and in-model positions of the array instances are those as if such arrays were created prior to scaling.
- ▼ The **mates** applied to assembly objects ignore any subsequent changes in object size and position as a result of the scaling. To correct mates, edit them by re-specifying mated objects.

2.3.8.8. Body or surface position change

Position change operation allows to move body or surface to an arbitrary model point and to rotate it at arbitrary angle.

The result of the operation is a **new object**— body or surface. Objects derived from the initial (“old”) body or surface remain connected to this body or surface. New body or surface can be used with no restrictions in any future operations, including the position change operation.

Depending on the setting made during operation execution, an initial object is deleted from the model or remain in it. If this object was used in mates, the mates are marked as faulty in case this object is deleted. To correct mates, edit them by re-specifying mated objects.

Body or surface position change is made in the coordinate system defined by a user. It becomes coordinate system for change position operation (for more on object coordinate system see Section 2.8.3.4.1 on p. 598).

The Body or Surface Position Change command is available if there is at least one body or surface in the model. You can change the position of one body or one surface in a single command call. Note that change position operation is applicable to bodies and surfaces created in the model itself only (not in any of its components).

To change a body or surface position perform the following actions.



1. Start the body or surface position change operation To do this, click the **Change Position** button on the model editing toolbar or call the **Operation — Change Position** command.
2. Specify the object (body or surface), whose position should be changed, in the Design Tree or in the model window.



The body icons and names are shown on the Tree if the model structure is enabled (see Section 2.1.2.2 on p. 106).

If a model contains one body or surface, this body or surface is selected automatically.

A selected object and coordinate system (by default— the current) in which position change will be performed are highlighted in the Design Tree and in the model window.

A special element is displayed in the model window to control the object's shift and rotation— **a ghost of the coordinate system**. Object position changing is performed by changing position of this ghost. Mutual position of the object and of the coordinate system's ghost does not change. At first, the ghost coincides with the selected coordinate system. You can move ghost along coordinate system axes, along its own axes and also rotate around its own axes.

3. If you need to use another coordinate system, specify it in the Design Tree.

If there is no suitable coordinate system in the model, create new local coordinate system. To do this, click the **LCS Construction** button on the Special Control Panel. LCS creation process will be started (see Section 2.8.3.5 on p. 601). Specify LCS position and click the **Create Object** button. The system will resume the changing the object position operation process, created LCS will appear in the Design Tree and will be automatically selected as coordinate system in which position changing is performed.



4. Specify whether to delete the initial body or surface. To do this, activate the desired switch in the **Mode** group:



- ▼ **Delete source objects** – the number of bodies (surfaces) in the model does not change, the source body (surface) is not displayed in the model window, but displayed as a subordinate object of the *Change Position* operation in the Design Tree.



- ▼ **Keep source objects** – the number of bodies (surfaces) in the model is increased by one, the initial body (surface) is displayed in the window and in the Design Tree at its previous position.

5. To specify position and orientation of coordinate system ghost, activate the **Parameters** tab of the Properties bar. Two methods of setting position and orientation are available. They are

similar to those used when creating a local coordinate system. To choose a method, expand the **Method** list and select the required row:

- ▼ **Respective to CS**— respective to the specified coordinate system (See Section 2.8.3.6.1 on p. 603),
- ▼ **By Object**— by object the properties of which allow uniquely defining coordinate system ghost position and orientation (see Section 2.8.3.6.2 on p. 605).

Shift and/or rotation of coordinate system ghost is shown on the object ghost: it is positioned in accordance with the new position of the coordinate system.



If you need to position an object with respect to a coordinate system (CS1, for example) in the same way it is positioned with respect to another coordinate system (CS2), specify CS2 as the coordinate system to be used for changing position while SC1 should be specified as an object which defines position and orientation of the coordinate system's ghost.

6. If necessary, change the direction and color of new body or surface. For that, use control elements on the **Properties** tab of the Property bar.



7. Click the **Create Object** button on the Special Control Panel to complete the operation.

Body or surface will change its position in the model window; an icon will appear in the Design Tree indicating the position change operation.

2.3.9. Round hole

Creation of a round hole is one of the body editing operations. KOMPAS-3D supports creating round holes of the following types:

- ▼ **Simple hole,**
- ▼ **Countersink hole,**
- ▼ **Counterbore hole,**
- ▼ **Hole with countersink and counterbore,**
- ▼ **Conical hole.**

Construction of round holes is described in Section 2.3.9.1 on p. 208.

Besides that, you can create a round hole by a sketch profile using the library of holes. For this purpose, use the **Hole from Library** command (see Section 2.3.9.2 on p. 215).

The group of commands for creating round holes of different types and the **Hole from Library** command can be found in the **Operations – Hole** menu.

Their shortcut buttons are located on the model editing toolbar (Fig. 2.3.52).

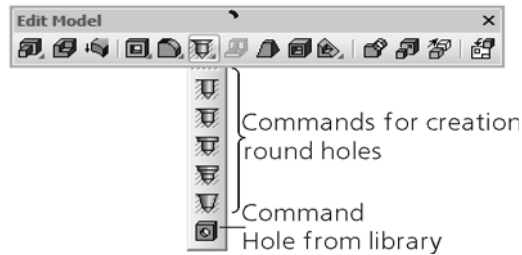


Fig. 2.3.52. Buttons for round hole creation commands and the **Hole from Library** command

2.3.9.1. Round hole: simple, with countersink/counterbore, conical



In order to create a certain type of hole, call the respective command:

Simple hole;



Hole with Countersink;



Hole with Counterbore;



Hole with Countersink and Counterbore;



Conical hole.

Once the command is called, control elements for setting the hole parameters appear in the Property Bar (Fig. 2.3.53). Some parameters are set for all types of holes, while other parameters depend on the hole's type.



During the construction process, you can change the hole's type by selecting the required line in the **Hole Type** list. Choosing a hole type from the list has the same effect as calling the command for creation of the respective round hole.

Sketch of the selected hole's type is displayed in the preview window on the **Hole Parameters** bar.

To create the round hole, complete the following steps.

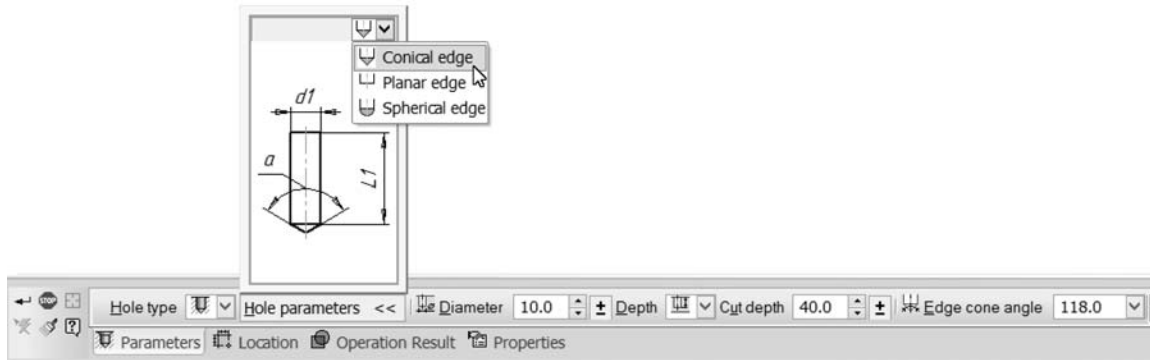


Fig. 2.3.53. Property bar during a simple hole construction

1. After calling the command, specify the surface on which the hole will be located. The ghost of the hole with the specified parameters will appear in the Model window. The hole's anchor point is the point specified for the hole on the selected surface.



During the construction process, you can choose another surface for the hole placement.

2. Enter the hole diameter value in the **Diameter** field.
3. Specify the hole's depth (see Section 2.3.9.1.1 on p. 210).
4. Specify parameters depending on the hole's type:
 - ▼ for a hole with countersink and/or counterbore (see Section 2.3.9.1.2 on p. 211),
 - ▼ for a conical hole (see Section 2.3.9.1.3).
5. If necessary, you can also:
 - ▼ Creating a threaded hole (see Section 2.3.9.1.4 on p. 212).
 - ▼ Specify the hole's edge form (see Section 2.3.9.1.5 on p. 213).
 - ▼ Creating hole axis. For that, enable the **Axis** option.
 - ▼ Specify the hole's location parameters (see Section 2.3.9.1.6 on p. 214):
 - ▼ position of the hole's anchor point,
 - ▼ slope and axis turn,
 - ▼ direction of the construction,
 - ▼ cutting out part of the body above the hole,
 - ▼ Creating a hole using a template (see Section 2.3.9.1.7 on p. 215).



Diameter, depth, and position of the hole on the surface can be changed using handles.

If you are working with a multi-body part, beside parameter setup, setting of operation application area can be necessary. For this, the **Operation Result** tab of the Properties panel is used. Detailed information on operation application area and methods of its specification is

provided in Section 2.3.10.3 on p. 219.



6. Upon completion of the hole parameterization click the **Create Object** button on Special Control Panel.

The model with the hole on the specified surface will be displayed in the window, and an icon corresponding to the selected type of hole will appear on the Design Tree. If the hole was created with an axis and/or thread, an additional axis and/or thread icon will appear on the Design Tree as a subordinate object of the hole. Next to its symbolic depiction, thread specification is given (for non-standard thread, a *Symbolic thread depiction* message is used).

2.3.9.1.1. Specifying the hole's depth

For all types of round hole, depth is specified starting from the hole's anchor point to the center of the edge of the hole's straight round cylinder.





The hole's depth can be specified using different methods.

Choose the method of specifying the hole's depth from the **Depth** list (see Table 2.3.14).



Depth of the hole having a conical or spherical edge is increased by the edge's height. For detailed description of the edge's form selection, refer to Section 2.3.9.1.5 on p. 213.

Tab. 2.3.14. Method of specifying the hole's depth




	Value of the Depth option	Building rules
	By Length	The hole's depth is defined by the specified distance. The Depth of cut field is available. Enter the distance's value in that field.
	Through All	The hole's depth is defined by distance between the specified surface and the intersection point of the hole's axis with the plane perpendicular to the axis and going through the point of the model's outline box which is the outermost from the anchor point.
	To Vertex	The hole's depth is defined by the absolute value of the distance between the hole's anchor point and the projection of the point object to the hole's axis. Specify the point object in the Model window. The name of the selected object is displayed in the Vertex field beside the switch. To change the point object, enable the Vertex switch and reselect the object.
	Up to Surface	Depth of the hole is defined automatically by position of a face or a plane specified by user. The hole is cut up to the selected surface. The endface of the hole has the surface form. Specify the surface in the model window that limits the hole's depth. The name of the selected object is displayed in the Surface field beside the switch. To change the object, enable the Surface switch and reselect the object.

2.3.9.1.2. Parameters of the hole with countersink and/or counterbore

For a hole with countersink and/or counterbore, further to parameters that are general to all types, countersink and/or counterbore setup parameters are also specified. All parameters are displayed in the hole's sketch in the preview window on the **Hole's Parameters** bar (see Fig. 2.3.53 on p. 209).

- ▼ For a hole with countersink, choose the countersink parameters' specification method from the **Method** list (see Table 2.3.15) and specify the respective values.

Tab. 2.3.15. Method of setting countersink parameters

	Value of option	Building rules
	Method	
	By diameter and angle	The countersink surface is defined by its diameter and cone angle. Enter the countersink diameter value in the Countersink diameter field. Specify or choose the value of the countersink cone angle from the Countersink angle list.
	By depth and angle	The countersink surface is defined by its depth and cone angle. Enter the countersink depth value in the Countersink depth field. Specify or choose the value of the countersink cone angle from the Countersink angle list.
	By diameter and depth	The countersink surface is defined by its depth. Enter the diameter value in the Countersink diameter field. Enter the depth value in the Countersink depth field.

- ▼ For a hole with counterbore:
 - ▼ specify the counterbore diameter in the **Counterbore diameter** field,
 - ▼ specify the counterbore depth in the **Counterbore depth** field,
- ▼ For a hole with countersink and counterbore:
 - ▼ specify total diameter of countersink/counterbore in the **Countersink/counterbore diameter** field,
 - ▼ specify the counterbore depth in the **Counterbore depth** field,
 - ▼ enter the counterbore cone angle in the **Counterbore angle** field.



All changes of dimensions will be reflected in the hole's ghost.

2.3.9.1.3. Conical hole parameters

For a conical hole, apart from parameters that are general for all types, conical form parameters should be set up. There are two methods to set the parameters. Using each of the methods, conical form of the hole is achieved by increasing the diameter of the cone's upper base (the diameter going through the anchor point of the hole). The diameter of the lower base of the cone is equal to the value specified in the **Diameter** field. All parameters of the conical hole are displayed in the hole's sketch in the preview window on the **Hole's Parameters** bar (see Fig. 2.3.53 on p. 209).

Once the cone's lower base diameter is set together with the hole's depth, choose the method of setting parameters of the conical hole from the **Method** list and specify their values (see Table 2.3.16).

Tab. 2.3.16. Method of setting parameters of a conical hole

	Value of option	Building rules
	Method	
	By diameter of the cone's upper base	The hole's surface is defined by the diameter of the upper base of the cone. Specify the diameter of the cone's upper base in the Base Diameter field.
	By cone's angle	The hole's surface is defined by the cone's angle. Specify the cone's angle in the Cone's angle field.

2.3.9.1.4. Creating a threaded hole

Round hole of any type can be threaded. In this case when creating the hole a symbolic depiction of thread is formed, too. Hole diameter is defined by thread parameters — nominal diameter and step. The thread can be both standard and non-standard.

To create a threaded hole take these steps.

1. Open the **Hole's Parameters** bar (see Fig. 2.3.53 on p. 209).
2. Click the **Threaded hole** button.
On the Properties Bar controls will appear for setting thread parameters.
3. Select the name of the thread from the **Thread standard** list. The list contains the names of standard threads and a **Non-standard thread** item.
4. Set thread parameters.

Thread parameters can be set using the following methods.

▼ Method 1

- ▼ For non-standard thread type or select from the list the value of nominal thread diameter in the **Thread diameter** field, and pitch—in the **Pitch** field.
- ▼ For standard thread, pitch value is set automatically after specifying the diameter. If the **Pitch** list contains several pitch values, you can change the automatically set value.

▼ Method 2 (for standard thread).



- ▼ Click the **Select standard thread parameters** button. A **Select standard thread parameters** dialog box with a table will appear on the screen.
The **Standard** list on the dialog box displays the name of selected thread standard. If required, you can select a different thread standard on the list.
The column of the table contain the values of nominal thread diameter, pitch and other thread parameters applicable to the selected standard.
- ▼ Highlight the appropriate row in the table and click **OK**. On the Properties panel, the **Thread diameter** field will show the value of nominal thread diameter, and the **Pitch** field—value pitch from the highlighted row.

5. Select the method of setting thread length in the **Thread length** group of switches.

The following methods are available.



- ▼ **Specified depth:** thread length is defined by the specified distance. Enter thread length value in the **Thread Length** field.

The object from which the distance is calculated is (Fig. 2.3.54):

- ▼ for countersink holes — lowest point of the countersink surface,
- ▼ for all other holes — hole location surface.

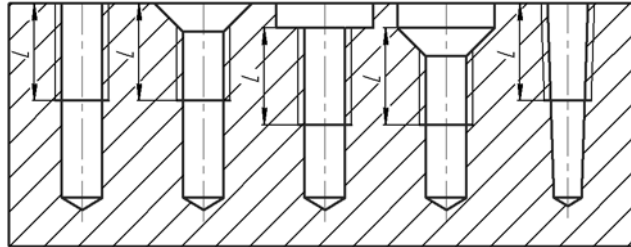


Fig. 2.3.54. Initial thread border for different types of holes



- ▼ **Total depth:** thread length is determined automatically using the depth of the hole.



If required, you can change the automatically calculated end limit of the thread. To do so, activate the **End limit** switch, then specify on the Design Tree the surface, facet, or plane for which you want to plot the hole.

Symbolic thread depiction appears on the hole ghost.



In a conical hole, standard thread can be plotted to a specified depth only. Maximum depth depends on the hole dimensions and is set by the standard.

Standard thread tables

Standard thread parameter tables are found in the folder whose path is set in the system variable *Thread* (see Section 9.5.4 on p. 1886) of the system KOMPAS-3D. By default, it is subfolder *...Sys\Thread* in the main system folder.

You can manage the list of standard thread tables and their connection using the **Thread parameters** (see Section 9.1.11.11 on p. 1736). The delivery package includes KOMPAS-3D metrical, pipe, buttress and trapezoidal thread tables.



The said tables contain standard thread parameters. Use them as templates for creating your own thread tables.

When a symbolic depiction of thread is created in the model using the tables, no link between the model and the thread table file is formed.

2.3.9.15. Specifying the hole's edge form

When creating a hole, you can define the form of its bottom.

Open the **Hole's Parameters** bar (see Fig. 2.3.53 on p. 209). Open the **Edge Form** list and choose the required variant.



▼ **Conical edge,**



▼ **Flat edge,**



▼ **Spherical edge.**

For a conical edge, specify the cone's angle value in the **Edge's cone angle** field.

2.3.9.1.6. Hole positioning on surface



There are several parameters of positioning a hole on surface. To set the parameters, activate the **Positioning** tab. The Property Bar will display controls for specifying the hole's position on surface, orientation of its angle and direction of its construction.



Position on surface

In the model window, specify the surface for positioning the hole making sure the **Surface** button is enabled. The name of the chosen object will be displayed in the **Placement Surface** field. Positioning of the hole's anchor point on the selected surface is identical to creation of a point on surface (see Section 2.5.2.6 on p. 339).

Direction of the hole's axis

By default, the hole's axis is perpendicular to the placement surface. You can change the direction of the hole's axis. To do this, turn off the option **Perpendicular to surface**.

There are several methods to change the direction of the hole's axis:

▼ Inclination and rotation of the hole's axis as compared to its default position.

In order to incline the hole's axis, specify inclination angle in the **Inclination angle** field.

In order to rotate the hole's axis, specify the rotation angle in the **Rotation angle** field.

▼ Specifying the guide object.



Enable the **Axis direction** switch and specify the guide object in the window or in the Design Tree. The guide object can be an existing straight-line, flat object or a rotation surface (except for sphere). The straight-line object sets direction parallel to itself; the flat object sets direction perpendicular to its plane, while the rotation surface sets direction which is parallel to its axis. The list of straight and flat objects is presented in Table 2.1.7 on p. 131.

▼ Creation of a guide vector



Direction of the hole's axis will coincide with the vector's direction.

In order to create the vector, use the **Vector Construction** button on the Special Control Panel. For more information about construction of the guide vector, see Section 2.8.5 on p. 613.

Building Direction

Direction of the hole's construction can be specified using switches of the **Direction of Construction** group. By default, the hole is built in the inside direction with respect to the surface



on which it is located. This direction is **Straight**. Activate the **Reverse Direction** switch to change the direction of construction.

Deletion of the body's part above the hole ("canopy" processing)

In some cases, the body, in which a hole is created, forms a "canopy" above it. The "canopy" can be removed by lengthening the hole's surface in the direction which is opposite to the hole's construction direction. For this purpose, the **Process "canopy"** option is designed (Fig. 2.3.55).



Fig. 2.3.55. Creation of a hole with countersink: a) the **Process canopy** option is disabled; a) the **Process canopy** option is enabled

Note that if no new face is formed immediately above the hole as a result of the hole construction, the body's material will not be removed, since in this case it is considered that there is no "canopy" (Fig. 2.3.56).

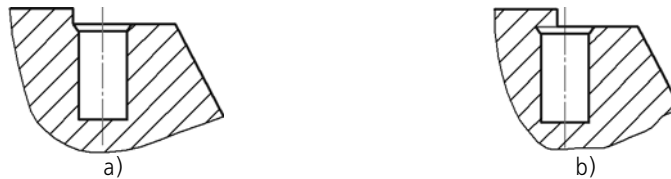


Fig. 2.3.56. Hole's position in the model: a) no "canopy", b) with "canopy"

2.3.9.1.7. Creating a hole using a template

To plot a hole you can use the parameters of a template hole.



On the Special Control Panel, click **Hole by Template** (the **Parameters** tab must be active on the Properties panel). Specify the hole to copy the parameters of in the model or on the Design Tree. The hole being created will have the same parameters, as the template hole. Edit the parameters, if required. If the position of the hole is not set yet, set it now.

2.3.9.2. Hole from Library



In order to create a round hole using the library of holes, select the flat object on which it should be positioned (the list of flat objects is presented in Table 2.1.7 on p. 131). Then, call the **Hole from Library** command.

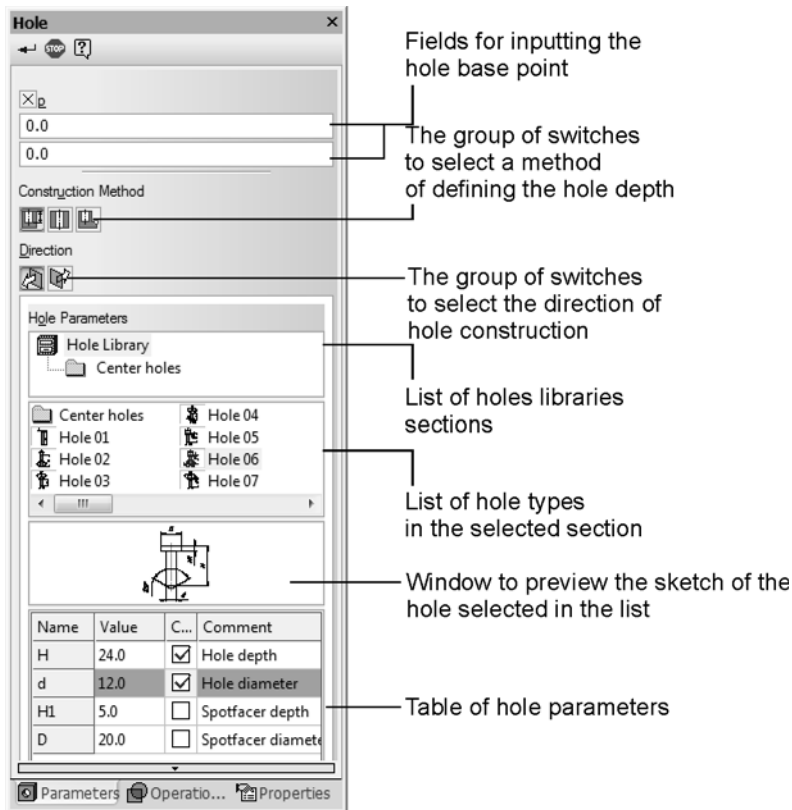


Fig. 2.3.57. Hole properties panel

Upon calling the command, control features to select the hole profile and to enter its geometric sizes shall appear (Fig.2.3.57) on Properties toolbar.

Select the type (shape) of the hole from the drop-down menu. The selected profile sketch of the hole type is displayed in the preview window below the list of types.

This sketch is parametric. The hole parameters are controlled by values of applicable sizes in the sketch. List of variable parameters of the hole is displayed below the preview window.

To change any of the hole parameters, enter the required value into corresponding field of parameter table.



Not all dimension values can be modified arbitrarily. For example, the thread diameter cannot be made greater than the nominal diameter of the hole. If you need to increase the hole diameter, first change the nominal diameter, and then the thread diameter.



Enter the required values of all hole parameters.

If the overall depth **H** parameter is present among parameters of the selected hole, the switch keys of the **Method of Construction** group become available. They allow to indicate which method the hole depth is defined with.

If the **On Depth** switch is activated, the depth of the hole shall be equal to the specified value in the list of parameters. If the **To Vertex** or **Through All** switch is activated, the H parameter disappears and the depth of the hole is defined automatically. The concept of automatic depth determination is the same as the one for cutting extrusion feature. If **To Vertex** switch is disabled, you should select this vertex.

The hole ghost with specified parameters will appear in the Part window. By default, the snap point of the hole (it is marked in red color in the sketch) is located in the origin of the local coordinate system of the face on which this hole is created.

To fix the hole in the required place of the face, unfix the fields for input of the snap point coordinates. For this purpose, click the m field. The cross in this field will change to a checkmark. Indicate the position of the hole with mouse or enter coordinates of the hole center.

If you are working with a multi-body part, beside parameter setup, setting of operation application area can be necessary. For this, the **Operation Result** tab of the Properties panel is used. Detailed information on operation application area and methods of its specification is provided in Section 2.3.10.3 on p. 219.



Upon specifying all the parameters of the hole, click the **Create object** button on Advanced Control Panel.



The part with the hole on the specified face shall be displayed in the window, and the hole icon shall appear in the Design Tree.



If the hole fully intersects the body, the operation shall result in a body made up of multiple pieces (see Section 2.3.11 on p. 224).

You may create your own library of holes or add holes to the system library (see Section 12.1.4.3 on p. 2045).

2.3.10. Application area of operation

2.3.10.1. General Information

Operation application area is a set of objects affected by the result of the operation. The application area of the operation can comprise the bodies or components the model contains, or both.

When you add model material (e.g. when you paste shape-generating features), the application area defines the bodies to be combined with the feature being created (edited); and when you delete model material (e.g. when you cut shape-generating features out), it defines the bodies which material will be deleted as a result of the operation.

Let's assume the model contains three bodies. An extrusion operation sketch is created on the top face of body 1 - a circle (Fig. 2.3.58, a). The method of defining the extrusion depth is **Through All**. Any of those bodies, and in any combination, can be included in the operation application area. Let us suppose that the application area consists of the bodies 1 and 3.

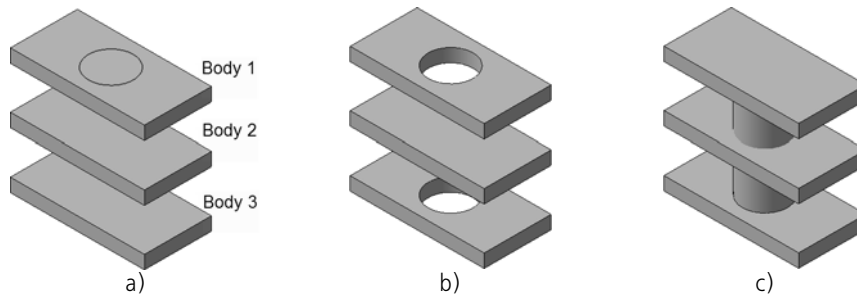


Fig. 2.3.58. Extrusion operation application area:
 a) initial state of part and sketch of operation,
 b) result of cutting, c) result of editing a boss

- ▼ If the extrusion feature is to be cut out, the operation will produce a hole only in the bodies 1 and 3 (Fig. 2.3.58, b). Even though the extrusion feature goes through the body 2, it will stay unaffected since it is not included in the operation application area.
- ▼ If the extrusion feature is to be pasted, then a new body consisting of the bodies 1, 3 and the extrusion feature will appear (Fig. 2.3.58, c). Even though the extrusion feature goes through the body 2, it will stay alone, since it is not included in the extrusion operation application area.

2.3.10.2. General procedure for setting operation application area

1. Selection of the application area type. To do this, the switch **Application Area** is used. It appears on the page **Parameters** of the Properties bar during execution of material adding or removal operation in the model (Fig. 2.3.59).



- ▼ Enable the switch **Application Area "Components and bodies"** so that the operation effect covers the components included in the assembly and also the bodies created in the model.



- ▼ Enable the switch **Application Area "Components"** so that the operation effect covers only the components of the model.



- ▼ Enable the switch **Application Area "Bodies"** so that the operation effect covers only the bodies of the model.

2. Definition of the application area of the selected type — specifying the objects which comprise the applicability range.

Definition of the “Body” application area for material removal operations can be found in Section 2.3.10.3 on p. 219, definition of the “Components” application area— in Section 2.3.10.4 on p. 222. If the application area “Components and Bodies” has been selected, then you should define both the areas— “Components” and “Bodies”.

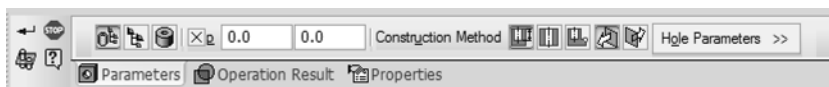


Fig. 2.3.59. The Properties bar during the creation of a round hole from library



When working with an assembly (*.a3d), the material removal operations have by default the 'Components and Bodies' application area enabled, and the material adding operations—'Bodies'. When working with a part (*.m3d) operations of both types have an application area of 'Components and Bodies'.



The default method to define the application areas 'Components' and 'Bodies' for all material removal operations, except for surface cut-off or sketch, — **Auto determination**. For surface and sketch cut-off operations the default method to define the 'Components' application area is **All Components**, and that of 'Bodies' application area—**Auto Define**.

2.3.10.3. "Body" Application Area

The "Body" application area may include:

- ▼ visible (non-hidden and included into calculation) bodies which intersect with the feature created due to operation,
- ▼ all shown bodies,
- ▼ arbitrary set of bodies.

By default, the application area of an operation is determined automatically: it includes all bodies except for the hidden ones intersecting with the operation-produced feature (it is shown as a ghost before completing the operation).



Thus, if you are working with intersecting bodies, we recommend hiding those of them that should not be included in the application area of the performed operations.

In the examples shown on Fig.2.3.58, the result of cutting with automatic application area determination would be the holes in all three bodies, while the result of pasting shall be a single body combining bodies 1, 2, 3 and the extrusion feature.

If necessary, the scope of any operation application may be set up manually, including all visible bodies or an arbitrary set of bodies.

Methods to define the application area are detailed in Section 2.3.10.3.3.

The set of bodies constituting the operation application area **remains unchanged**. Enabling the display of bodies that were hidden at the time of performing the operation as well as "dragging" bodies in the Design Tree do not affect the application area. Due to this, modeling parts by subtracting one body from another is possible.

For example, you need to construct a part with a complex-shape void. To do this, you can do the following.

1. Create the first body, i.e. the body that will define the part shape, and then hide it.
2. Create the second body, i.e., the body that will define the void shape. While executing operations to create the body, it is possible not to carry out any actions to define their application area, i.e. you may use the default method to define the application area — automatic determination. In this case, only the second body will be included in the application area of all operations.

3. Enable the first body display. This will not change the application area of the second body. In other words, all operations performed in step 2, including, for example, the **Through All** cut operations, will still affect only the second body.
4. Perform the Boolean operation of subtracting the second body from the first one.

To include new bodies in operation application area or exclude existing ones from it, switch to the editing mode of the operation and modify its application area using the control features on the Properties toolbar. These features and methods of working with them are described in the further sections.

2.3.10.3.1. Determination of the "Body" Application Area for Material Adding Operations

While adding material to a model, the 'Body' application area is specified on the **Operation result** tab of the Property panel (Fig. 2.3.60).

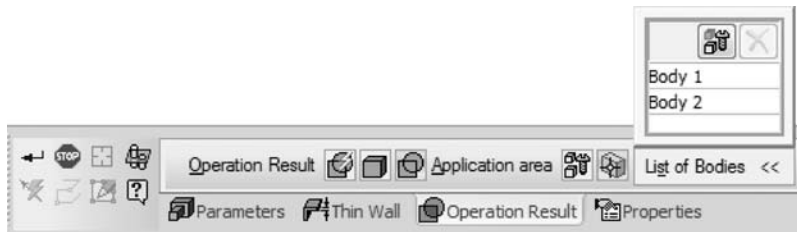


Fig. 2.3.60. Setting 'Bodies' application area for pasted extrusion element



The default method of defining an operation application area is the auto determination. The **Auto Union** switch in **Operation Result** group is active in this method.



If necessary, you can set up the operation applicability of range in an arbitrary way. To do this, activate the **Union** switch. The **Application Area** group of features becomes accessible on Properties toolbar.

Methods to define the application area are detailed in Section 2.3.10.3.3.

2.3.10.3.2. Determination of the "Body" Application Area for Material Deleting Operations

- ▼ While cutting shape-generating features, the "Body" application area is specified on **Cutting** tab of Properties toolbar (Fig.2.3.61).
- ▼ While creating a circular hole, as well as at dissection of the model with plane or sketch, the "Body" application area is specified on **Operation Result** tab of Properties toolbar (Fig.2.3.62).

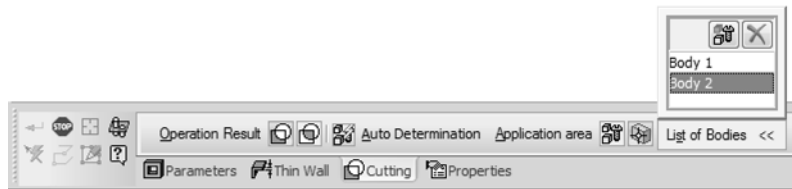


Fig. 2.3.61. Setting 'Bodies' application area for extrusion element being cut

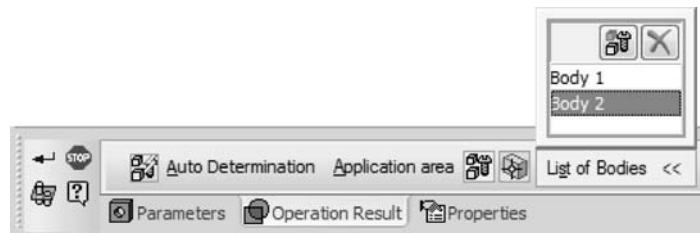


Fig. 2.3.62. Setting 'Bodies' application area for a library hole



The default method of defining an operation application area is the auto determination. In this case, the **Auto Determination** switch is active.

If necessary, you can set up the operation applicability of range in an arbitrary way. To do this, turn off the **Auto Determination** switch. The **Application Area** group of features becomes accessible on Properties toolbar.

Methods to define the application area are detailed in Section 2.3.10.3.3.

2.3.10.3.3. Methods to define the "Body" application area



Auto Determination of the application area indicates inclusion of all bodies, except for the hidden ones and the ones excluded from calculations, intersecting with the feature created as the result of the operation.

- ▼ If no intersections were found between the feature being created and other bodies when adding material to the part, then the feature is created as a stand-alone body.



In this case, the **New Body** switch activates on the **Operation Result** tab.

- ▼ If, upon removing material from the part, no intersections were found between the feature being created and other bodies, the error occurs in the model.



Automatic determination of the operation application area is in a sense preferable to other methods of defining the application area since in the latter case the bodies that do not intersect with the feature being edited may accidentally become included. In this case, the feature cannot be created. The reason for this is that the operation can be performed only if the feature resulting from the operation does actually intersect with the bodies that enter its application area. If the feature does not intersect with at least one of the bodies included in the operation application area, "The objects do not intersect" error occurs in the model.

When auto determination of the application area is disabled, the **Application Area** group of features becomes accessible. It contains two switches: **All Bodies** and **Selection of Bodies**.



- ▼ The **All Bodies** switch serves to include all bodies except for hidden and excluded from calculation, regardless of whether they intersect with the edited feature, or not.



- ▼ **Body Selection** switch serves to specify the bodies that shall be included in the current operation application area. Upon activating this switch, the **List of Bodies** toolbar becomes available. It contains the list of bodies included in the operation application area, and two buttons: **Select All** and **Delete**. To manually add bodies to the application area, select them in the model window or in the Design Tree.



The **Select All** button serves to include all bodies present in the model in the operation application area, including hidden bodies and bodies that do not intersect with the feature being edited.



The **Delete** button serves to exclude body from the application area.



The specific feature of **Auto Determination** and **All Bodies** switches is their functioning only during operation execution. The result of their work is creating corresponding lists of bodies. To verify this, enter the editing mode of the feature created with the **Auto Determination** or **All Bodies** switches disabled. You will see that this switch is off on the Property bar, the **Manual Selection of Bodies** switch is on, and the **List of Bodies** bar contains names of intersected bodies or of all bodies respectively.



The cutting operation (with a plane or by sketch) results in cutting of the bodies which intersect the cut surface when the **Auto Determination** switch is active.

If you need to delete the bodies entirely located on the same side of the cut surface in addition to cut body pieces, use the **All Bodies** switch or specify bodies manually.

2.3.10.4. "Components" Application Area

The "Components" application area may include:

- ▼ those of visible (non-hidden and not excluded from calculation) model components which are intersected by the element created as a result of the operation,
- ▼ all model components,
- ▼ all components except those from library,
- ▼ arbitrarily selected components.

2.3.10.4.1. Setup of Application Area "Components"



To go over to the mode of setting the 'Components' applicability area while performing the operation in the model, click the button **Application Area** on the Advanced Control Panel (Fig. 2.3.59).

On the Property Bar, the elements, enabling selection of the components to apply the current operation to, will appear (Fig. 2.3.63).

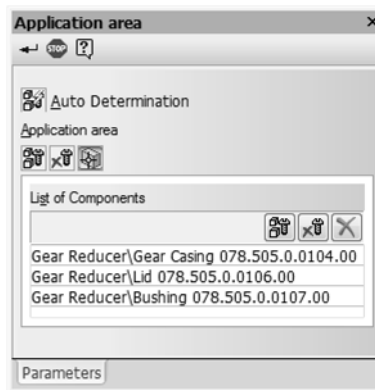


Fig. 2.3.63. The elements to control the application area “Components”



- ▼ To form the application area automatically, activate the **Auto Determination** switch. All the components, except hidden and excluded from calculation, intersecting with the element, being the result of operation (it is shown in the model window as the ghost), will be included into the application area.

A specifics of automatic determination of the application area is that it works only at the time of performing the operation. Its result is a created list of bodies with which an intersection is found when performing the operation. To witness that, go into the editing mode of a feature created using automatic determination of the application area. You will see that the automatic application area determination switch on the Property bar is disabled, the **Manual Selection of Bodies** switch is enabled, and the **List of components** bar contains the names of elements belonging to the bodies, with which there are intersections.

- ▼ To setup the application area in an arbitrary way, disable the switch **Auto Define**. The **Application Area** group of features becomes accessible on Properties toolbar.



- ▼ Enabling the switch **All components** means that the components, which intersect with the edited element, will be found and included into the area of operation application. However, the list of components, unlike auto defining the application area, is not created. Later on, when adding into the model the new components or moving the existing ones, the operation application area will be redefined and, after the model is rebuilt, the shape of components, included into the application area, will change.



- ▼ Enabling of the switch **All Components, save the library ones**, means the same as above procedure of defining the application area, except, however, for the objects inserted from the application libraries (for example, the fasteners from the Library of Standard Items).



- ▼ Enabling of the switch **Selected Components** allows manual selection of the components involved in the operation. To select the components, the panel **List of Components** is used (see Section 2.3.10.4.2). The area of the operation application set in this way is not changed at adding or displacing of the components. To exclude any components from the area of the operation application or add certain components, it is necessary to edit this operation by changing the earlier created list of components.



When finished with defining the application area, click the **Create Object** button on the Special Control Panel. The system will then return to the mode of executing the operation for which the application area “Components” was set up.

2.3.10.4.2. Generating the “Components” Application Area Manually

When defining the 'Components' application area manually, you can select any components in the model:

- ▼ parts,
- ▼ subassemblies,
- ▼ library components,
- ▼ copies of components within array instances.

Components can be selected in the Design Tree or in the Model window. The selected components are highlighted and their names are added to the **List of Components** panel (Fig. 2.3.63). A repeated selection of a component excludes it from the operation application area. If you need to exclude from the application area all the included in it components at once, click by the mouse in any free place of the model window.



A selected subassembly is always included in the operation application area together with all its components. Therefore, if you need to include only selected components of a subassembly into the application area, you need to select just those, watching that the entire subassembly is not selected.

To speed up component selection for the operation application area, you can use the buttons located on the **List of Components** panel.



After clicking the button **Select all**, all model components are included the operation application area: parts, subassemblies and copies of components belonging to array instances. This button is convenient to use if most of components are to be involved in the operation.



After clicking the button **Select all, except library ones**, all model components are included the operation application area, except those inserted from application libraries (*.rtw). If necessary, you can include some of library objects into the application area. To do this, pick them by the mouse in the Design Tree.



The **Delete** button allows to exclude from the application area the components whose names are selected in the list.

Note that an object selected in the list is highlighted in the Model window. This helps control correctness of component selection.

2.3.11. Bodies consisting of separate pieces

Solids in KOMPAS-3D models can consist of multiple disjoint pieces. For example, such a solid can be produced by extruding a sketch containing multiple contours.

It is also possible to divide solids into pieces as a result of cut or trim operation. For example, after inserting a base into a part, you can cut material out of it in a way to leave only those pieces of the base that are needed to construct the part.

There are special icons in the Design Tree that symbolize the presence of solids consisting of multiple pieces (Fig. 2.3.64):

- ! ▼ exclamation sign on the model icon signifies that the model contains solids made up of multiple separate pieces,
- ! ▼ exclamation sign on the operation icon signifies that the process of changing the set of pieces of the solid has been initiated during the operation (see Section 2.3.11.1),
- ▼ a solid icon divided into pieces signifies that the solid is made up of multiple separate pieces.



The icons of solids are displayed in the Tree, if model structure display is enabled in the Tree (see Section 2.1.2.2).

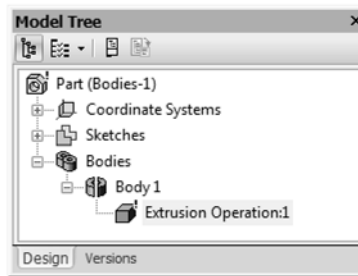


Fig. 2.3.64. The Design Tree of a part containing a solid made up of multiple pieces



The contextual menu, called in the Tree on the icon of a model or a body with the attributes of containing several parts, as well as the contextual menu called in the model window with bodies comprising parts, contain the **Attention** command. After calling this command, a message on presence of bodies consisting of multiple pieces appears on the screen.

Solids consisting of pieces can be made whole applying either of the two methods:

1. Change sketches or operation options that form solids out of pieces in a way that each solid should contain once piece. In this case, all signs of the presence of solids consisting of multiple pieces shall disappear from the Design Tree.
2. Create any new features connecting pieces of solids. In this case, only those signs of the presence of multiple pieces that are related to solids and models shall disappear from the Design Tree, while exclamations on icons of operations shall stay.



In the sheet body made up of multiple pieces, creation of new bends, changes in the state of current bends, or capping of bends are impossible. To perform these operations, a body shall be made single-piece.

2.3.11.1. Selecting pieces that need to be kept

Selection of pieces of a body (or bodies) that need to be kept is performed during the process of changing the set of pieces. This process is started automatically after completing the operation that leads to creation of a body consisting of multiple pieces.



Note that changing a set of parts is possible for bodies constructed in the model itself and impossible for component bodies. In other words, if operation separates component bodies into parts, all these parts remain.

By default, after the start of the process of changing the set of pieces, there is only one option enabled of the Properties Bar: **All**. It means that all the created pieces of the body shall be kept in the model. Body pieces are displayed in the Model window as ghosts.

If you do not need to keep all body pieces but only some of them, disable the option **All**. **List of Parts** (Fig.2.3.65) shall appear on the Property Bar. The list appears as a table, the first column of which contains the number of the body and pieces and the second column contains the option **Keep**.

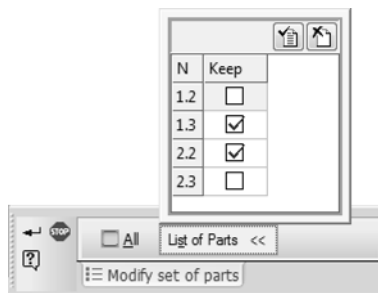


Fig. 2.3.65. Selecting body pieces that need to be kept



Disable options of the parts which will not be used, or specify these parts in the model window. Repeated specifying of the part will include it in the kept ones.



If you need to keep or exclude the majority of pieces, use buttons **Select All Parts** and **Deselect All Parts** that are located above the list.



Having finished the selection of the body parts, click the **Create Object** button. The button is not available if no pieces are kept.



Excluded body parts are as if they are not available in the model: they are not displayed, cannot be used for construction of other objects, not accounted for the MP calculation, not sent to associative model views and resulting files during export to other formats etc.

At the same time information about excluded parts is not deleted from the model file, allowing to change the set of parts in the future.

The process of changing the set of pieces is also initiated after editing the operation that shall result in a body made up of multiple pieces (unless the operation options were changed to result in a solid body).

The icon of the operation after executing or editing of which the process of selecting pieces has been started is marked with exclamation in the Tree. The **Modify Set of Parts** command is present in the context menu of this operation (Fig.2.3.66). It allows restarting the process of selecting pieces resulting from the operation.

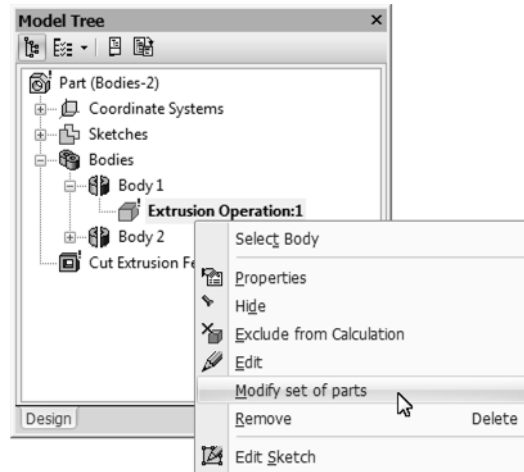


Fig. 2.3.66. Command **Modify Set of Parts**

2.3.11.1.1. Peculiarities of changing the number of the body's parts

1. If the process of changing the set of parts was launched for an operation, and then the number of parts was changed without launching the process, the message about the change in the number of parts will appear on the screen. For example, a sketch containing several contours was extruded with outside inclination. In the process of changing the set of parts, the user chose to retain only some of the body's parts. Then, in the Window of Variables, the value of the variable corresponding to the inclination level was increased so that some of its parts merged. After rebuilding, all parts of the body will be displayed in the model window, and the message about changing the number of parts will appear. If necessary, launch the process of changing the set of the body's parts and choose the required parts.
2. The operation is marked with exclamation only after starting the process of changing the set of pieces. Consequently, the following situation may occur.
The initial result of the operation was an intact body. After editing any object in the model, the operation resulted in a body consisting of multiple pieces. The model and the body receive an attribute of presence of multiple pieces, while the operation does not receive such an attribute because the process of changing the set of pieces was not started. To start the process, you need to enter the operation editing mode and then confirm the creation of an object.

Example 1

Extrusion of a sketch containing a single contour was performed in the model. Then the sketch was edited: a new contour was added into the sketch. Immediately upon exiting the sketch mode, the extrusion feature is displayed in the Model window as consisting of two pieces but the icon of the extrusion feature is displayed in the Tree without exclamation.

Example 2

The model contained a cut extrusion feature that deleted a certain volume of body material but

did not divide it into two pieces. Later in the Variables Window, the value of the variable that corresponds to the depth of extrusion was increased in a way that the cut feature became to fully intersect the body. After rebuilding, the body is displayed in the Model window as consisting of two pieces but the icon of the cut extrusion feature is displayed in the Tree without exclamation.

2.3.12. Multibody Modeling

Multibody modelling is the process of modeling when several bodies are created.

Multibody modelling may result either in one body or in multiple bodies (the case of a multibody model).

Each body during multibody modeling is created in a usual way. Boolean operations can be performed with bodies (see Section 2.3.7 on p. 185).

While executing operations in a multibody model, it is necessary to take into account their scope of application (see Section 2.3.10 on p. 217).

For each body, you can set MP parameters (see Section 2.14.6 on p. 802), the color and surface properties (see Section 2.14.2 on p. 784).

Multibody modeling expands the capabilities of constructing models and lifts the limitation on creating models that can be obtained solely by uniting, subtracting or intersecting bodies. For example, by using multibody modeling, a part may be constructed by combining thin-wall bodies with different wall thickness.

If necessary, the bodies constructed in the model can be saved as individual parts (see Section 2.11.2.3 on p. 689).



Note that body can not be displayed partially. That means that you cannot hide or show, for example, an individual feature pasted to the body. Therefore, after calling the **Hide** or **Show** command on some feature, the entire body which includes the feature gets hidden or shown.

2.3.12.1. Examples of Multibody Modeling Use

Among most typical examples of using multibody modeling are the “multilateral” design approach and creating bodies by subtraction.

The “multilateral” design is a design method in which isolated pieces of a part are created as stand-alone bodies which are merged subsequently. In this way, the intersecting bodies can be united by performing a Boolean operation on them, whereas the non-intersecting ones may be combined by constructing a new body that intersects with them.

Creating bodies by subtraction is the design method in which a body part is formed by subtracting some bodies from others. In this case, you need to create two intersecting bodies the first of which will define the part shape, and the second one— the shape of the void. After that, you will need to perform Boolean subtraction operation on the bodies, subtracting the second body from the first one. This design approach is recommended when creating parts with complex pockets or other cavities made through complex surfaces and using fillets.



It is impossible to copy boolean operation result using the array-creating commands. Therefore, if there should be several identical pockets or cavities resulting from subtraction, you should at first create copies of bodies for subtraction, and then execute boolean operation as many times as required.

2.3.12.2. Displaying a Multibody Model in the Design Tree

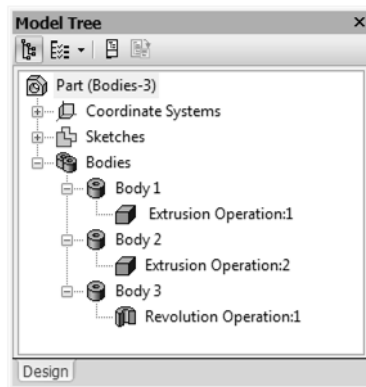
Overall amount of model bodies is displayed in the Design Tree in brackets after its name (Fig. 2.3.67–2.3.69).



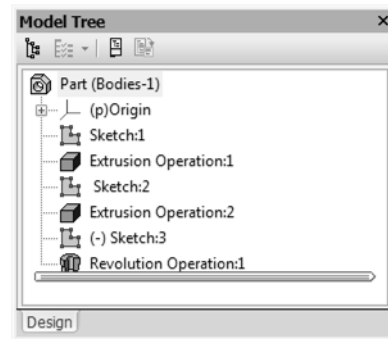
The number of bodies created in this model is displayed in the Design Tree. Number of bodies in components is not displayed.

The Design Tree can be displayed in one of the two types: as a sequence of construction or in form of a model structure (see Sections 2.1.2.1 on p. 106 and 2.1.2.2 on p. 106 accordingly). While working with a multibody part, representation of Tree as a structure is recommended. Rather than displaying of construction order, this method shall provide a better representation of multibody model content and the sequence of its construction (Fig.2.3.67–2.3.69).

- ▼ Body-forming features are grouped into categories that correspond to bodies. Sections have names “Body N”, where N is the sequential number of body (Fig.2.3.67,a).



a)



b)

Fig. 2.3.67. Multibody part in Tree:
a) structure, b) sequence of construction

- ▼ If the model undergoes a Boolean operation over bodies, then it is placed in a category matching the first of the involved bodies. The second body involved in this Boolean operation is displayed as being subject to it (Fig.2.3.68,a).

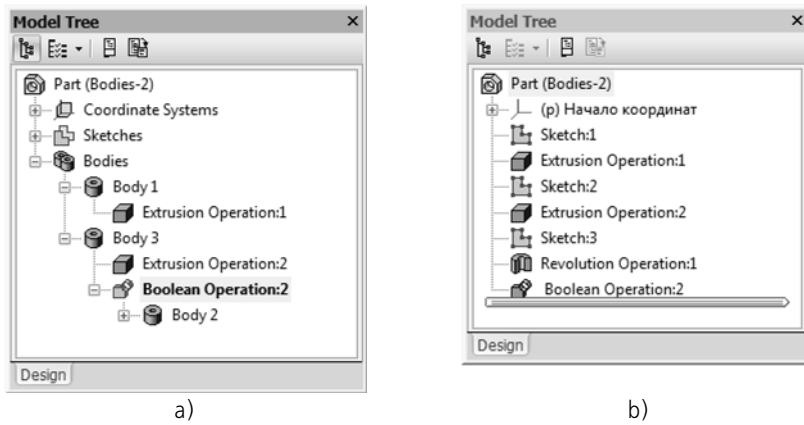


Fig. 2.3.68. Boolean operation over the part bodies:
 a) structure, b) sequence of construction

- ▼ If the model contains a feature combining several bodies into one, then it is placed into a category matching the first of the combined bodies. Other bodies merged through this feature are displayed as subject to it.

Combining features are represented in Tree with special icons (Fig.2.3.69,a).

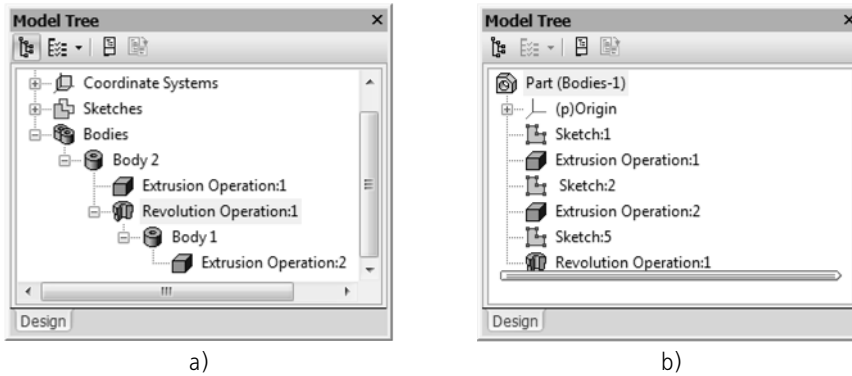


Fig. 2.3.69. Boss feature, combining bodies:
 a) structure, b) sequence of construction

2.4. Sheet bodies

2.4.1. General Information on Sheet Bodies

KOMPAS-3D supports modeling parts created from a sheet material by bending.

Commands for working with sheet bodies are located in the **Operations** menu, and the buttons to call them are on the **Features of Sheet Body** bar (Fig.2.4.1).



Fig. 2.4.1. **Features of Sheet Body** toolbar



Creating a sheet body creation starts with creating a sheet body. The **Sheet body** command (see Section 2.4.2 on p. 243) or **Shell** (see Section 2.4.3 on p. 246) may be used for that. Then elements of sheet body are added to the received sheet body: bends, plates, holes, cuts, etc.



Please note that sheet bodies can be added only to sheet body, built directly in the model (not in any of its components).

A sheet body is a sheet body together with added sheet body features. Bosses of features of any type— extrusion, revolved, sweep, loft — can be attached to a sheet model or cut from such model. Additionally, you can add design elements to a sheet body (fillets, chamfers, ribs, holes, etc.). Any copying operations can also be applied to sheet bodies, plates, holes and cuts.

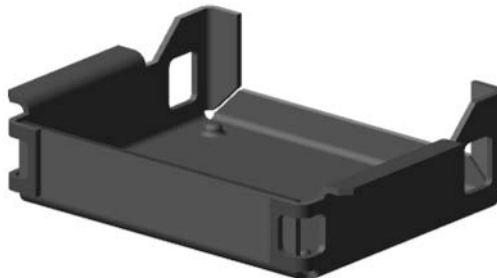


Fig. 2.4.2. Example of a part made of sheet material

One model can contain one or several sheet bodies. Multibody modelling is detailed in Section 2.3.11 on p. 224. If there are several sheet bodies in the model, they can have the same or different thicknesses.

2.4.1.1. Methods to Work with a sheet body

Any bend existing in a sheet body, can be displayed both in bent and rectified state. The user can at any time change the state of any bend (or a group of bends).

While working with a sheet body, it is possible to display its flat pattern. This is a special display mode, in which the bends selected by the user are shown as bent, while the rest are unbent. If neither of bends is selected, this mode displays a fully unbent sheet body. Switching to the flat pattern mode is possible after setting **flat pattern parameters** — selecting the bends that should stay bent.

Associative views of a model containing a sheet body are created on a drawing in the same way as associative views of a regular body. At the same time, if flat pattern parameters are specified in a sheet body, then it is possible to generate the flat pattern image of that part in the drawing.

If one model has several intersecting sheet bodies, then for their union, subtraction or intersection the **Boolean operation** command shall be used (see Section 2.3.7 on p. 185).

Stamped elements and elements with bends can be created only on those sections of a sheet body, which have the constant thickness (a sheet body with variable thickness can be created, for instance, as a result of uniting of two bodies with different thicknesses).

Please note that “non-sheet” operations in a sheet body may limit or exclude usage of the functional while working with sheet bodies:

- ▼ Some geometry-producing and additional structural elements may prevent changes of bends (see Section 2.4.7.3 on p. 307).
- ▼ Cut and trim operations may result in dividing a sheet body into parts. In each sheet body, which consists of several parts, it is impossible to create new and change existing bends, bends capping and create holes and cuts using the **Through** option. To perform these operations, a body shall be made single-piece.
- ▼ After the scaling operation, a sheet body becomes a “non-sheet” body: it is impossible to add any sheet elements to it or change bend states.

2.4.1.2. sheet body Parameters

A sheet body is characterized by the **thickness of the material** (S) of which it is made.

The bent portions (**bends**) of a part are defined by:

- ▼ **Inner radius** (R),
- ▼ **Bend angle** (α),
- ▼ **Width of bend release** (W),
- ▼ **Depth of bend release** (H).

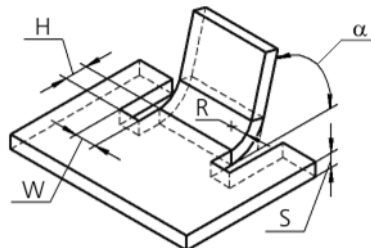


Fig. 2.4.3. Sheet Body Parameters



A bend may not have any releases.

Besides, any bend has a parameter that defines the flat pattern length of that bend. Depending on the selected method of defining the flat pattern length, this parameter can be the **Neutral Layer Factor**, or the **Bend Amount**, or the **Bend Reduction** (see Section 2.4.1.4 on p. 237). The parameter that defines length of flat pattern of bend, will be further referred to as **flat pattern parameter** of this bend.

Default parameters for all new sheet bodies are defined in the dialog of sheet body properties (See Section 9.2.7.4.8 on p. 1851).



To set up such parameters of operation as sheet thickness, bend width, slope and others, you can use handles (handles are detailed in Section 2.1.4.2.2 on p. 138).

2.4.1.3. Sheet Body Variables and Work with Them

While creating a sheet body in a model (see Section 2.4.2 on p. 243), **sheet body variables** are automatically created in it. Initial values of variables are defined by settings (see Section 9.2.7.4.8 on p. 1851).

Sheet body parameters and their current values are displayed in the Variables window (Fig. 2.4.4):

- ▼ SM_Thickness — sheet thickness,
- ▼ SI_Radius — inner bend radius,
- ▼ SM_Angle — bend angle,
- ▼ SM_BA — bend amount,
- ▼ SM_BD — bend deduction (reduction),
- ▼ SM_K — position coefficient of the neutral layer,
- ▼ SM_H — depth of bend release,
- ▼ SM_W — width of bend release.

Name	Expression	Value	Parameter	Comment
Part (Bodies-1)				
SM_Thickness	1.0	1.0		Sheet body thickness
SM_Radius	5.0	5.0		Bend radius
SM_K	0.40	0.40		Coefficient
SM_BA	10.0	10.0		Bend amount
SM_BD	0.0	0.0		Reduce bend
SM_Angle	90.0	90.0		Bend angle
SM_H	0.0	0.0		Depth of bend relief
SM_W	3.0	3.0		Width of bend relief
(p)Origin				
(-) Sketch:1				
Sheet Body:1				
v26		0.0	Exclude from Calculation	
v27	SM_Thick...	1.0	Sheet body thickness	
v37	SM_Radius	5.0	Bend radius	
v38	SM_BA	10.0	Bend amount	
v39	SM_BD	0.0	Reduce bend	
v40	SM_K	0.40	Coefficient	
v34	SM_Angle	90.0	Bend angle	
v35	SM_H	0.0	Depth of bend relief	

Fig. 2.4.4. Sheet Body Variables

By default, when creating elements with bends and the *Sheet body* element, the parameter values coinciding with values of the corresponding variables are suggested. If this values are **not changed manually**, then after creation of an element the following variables will be automatically assigned to its parameters:

- ▼ *Sheet Body* element:
 - ▼ Thickness = SM_Thickness;
- ▼ Elements with bends
 - ▼ Inner radius= SM_Radius,
 - ▼ Bend angle= SM_Angle,
 - ▼ Release depth= SM_H,
 - ▼ Release width= SM_W,
 - ▼ Bend amount= SM_BA,
 - ▼ Reduction of bend= SM_BD,
 - ▼ Neutral layer coefficient= SM_K.

New variable values of a sheet body in the current model can be defined in the Variables window or when editing a sheet body, to parameters of which variables were automatically assigned (to change the variable value it is required to change the parameter value, e.g. to increase the sheet material thickness).

The changed value of the variable, that is used by different elements, is transferred to all those elements. Thus, you can quickly change dimensions and topology of a sheet body without editing each element individually.

For instance, parameter “radius” in all part bends refers to the variable SM_Radius, and parameter “angle” — to the variable SM_Angle. Initial values of variables— 3 and 90 (Fig.2.4.5).

Variables				
Name	Expression	Value	Param...	Comment
Part (Bodies-1)				
SM_Thic...	1.0	1.0		Sheet body thickness
SM_Radi...	3.0	3.0		Bend radius
SM_K	0.40	0.40		Coefficient
SM_BA	10.0	10.0		Bend amount
SM_BD	0.0	0.0		Reduce bend
SM_Angle	90.0	90.0		Bend angle
SM_H	0.0	0.0		Depth of bend relief
SM_W	3.0	3.0		Width of bend relief
(p)Origin				



Fig. 2.4.5. Initial values of variables and source state of a part

Resulting from change of value of the variable SM_Radius by 1.5, and the variable SM_Angle by 70, all the part bends were rebuilt in accordance with new parameter values (Fig.2.4.6).

Variables				
Name	Expression	Value	Param...	Comment
Part (Bodies-1)				
SM_Thic...	1.0	1.0		Sheet body thickness
SM_Radi...	1.50	1.50		Bend radius
SM_K	0.40	0.40		Coefficient
SM_BA	10.0	10.0		Bend amount
SM_BD	0.0	0.0		Reduce bend
SM_Angle	70.0	70.0		Bend angle
SM_H	0.0	0.0		Depth of bend relief
SM_W	3.0	3.0		Width of bend relief
(p)Origin				



Fig. 2.4.6. New values of variables and a rebuilt part

The automatically created variables can be used in expressions defining the values of other variables and parameters, as well as in sketches.

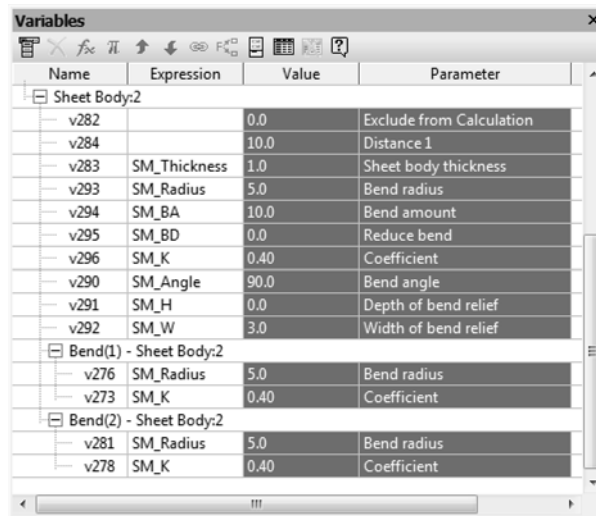


Work with variables and expressions is described in detail in Section 7.1 on p. 1569.

2.4.1.3.1. Peculiarities of Work with Sheet Body Variables

1. It is impossible to change default variables of a sheet body.
2. Editing of a bend, to parameters of which variables were automatically assigned, by changing the value of any parameter **does not affect** the value of the corresponding variable. Resulting from such editing, the bend parameter is assigned a value entered in the corresponding field of the Properties panel, while equality of parameter and variable values is automatically deleted.

3. If a value corresponding to the variable is returned to the parameter as a result of bend editing, then this variable is automatically assigned to this parameter.
4. You can create variables (SM_Thickness, SM_Radius, SM_Angle, etc.) in a model and assign the right values to them even before building a sheet body. The defined values are transferred to the dialog of setting up parameters of a sheet body for the current model and used when creating sheet elements.
5. Bends that have switched off definition of parameters based on the source object (see Section 2.4.4.3.6 on p. 260), are displayed in the Variables window as independent elements subordinate to their source object (Fig. 2.4.7). Parameters of these bends are radius and flat pattern parameter. If the parameter value coincides with the value of the corresponding variable, then this variable is automatically assigned to this parameter:
 - ▼ radius— SM_Radius,
 - ▼ bend amount— SM_BA,
 - ▼ bend deduction (reduction)— SM_BD,
 - ▼ neutral layer coefficient— SM_K.



Name	Expression	Value	Parameter
Sheet Body:2			
v282		0.0	Exclude from Calculation
v284		10.0	Distance 1
v283	SM_Thickness	1.0	Sheet body thickness
v293	SM_Radius	5.0	Bend radius
v294	SM_BA	10.0	Bend amount
v295	SM_BD	0.0	Reduce bend
v296	SM_K	0.40	Coefficient
v290	SM_Angle	90.0	Bend angle
v291	SM_H	0.0	Depth of bend relief
v292	SM_W	3.0	Width of bend relief
Bend(1) - Sheet Body:2			
v276	SM_Radius	5.0	Bend radius
v273	SM_K	0.40	Coefficient
Bend(2) - Sheet Body:2			
v281	SM_Radius	5.0	Bend radius
v278	SM_K	0.40	Coefficient

Fig. 2.4.7. Displaying of bends in the Variables window



If length of the flat pattern of the bend is defined using the table, the flat pattern parameter of this bend is not displayed in panel **Variables**.



If a bend belongs to a sheet body with an open sketch and corresponds to an arc in the sketch of the sheet body, then the *Radius* parameter of that bend does not show in the **Variables** window. And if length of the flat pattern of such a bend is defined using the table, the bend is not displayed in panel **Variables**.

2.4.1.4. Length of the Flat Pattern of a Bend

Length of the flat pattern of the bend can be defined using one of three methods:

- ▼ defining **Position coefficient of the neutral layer**,
- ▼ defining **Bend value**,
- ▼ defining **Bend Reduction**.

Besides, it is possible to use **Tables of bends**, i.e. extraction of the parameter that defines length of the flat pattern (bend value, bend reduction or coefficient of neutral layer), from the table.

The default method of defining a flat pattern length is set in the sheet body properties dialog for new models. For each specific model the default method to define flat pattern length can be changed in the sheet body properties setup dialog for the current model (for setup of default parameters for sheet bodies see Section 9.2.7.4.8 on p. 1851).

Besides that, it is possible to choose the method of defining the flat pattern length for each particular bend (see Section 2.4.4.3.5 on p. 259).

2.4.1.4.1. Definition of Flat Pattern Length Using the Position Coefficient of the Neutral Layer

Flat pattern length is determined on the assumption that there is a **neutral layer** in a bend.

Neutral layer is a part material layer, the length of which is not changed by bending.

The intersection line between the neutral layer of a bend and the plane perpendicular to the bend line is called the **neutral line** (Fig 2.4.8).

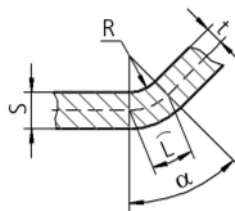


Fig. 2.4.8. Incision section by a plane perpendicular to the bend line
(the neutral line is shown by strokes)

The flat pattern length of the cylindrical portion of a bend is defined as the length of the **neutral line** in it:

$$L = \pi \cdot (R + K \cdot S) \cdot (\alpha/180), \text{ where}$$

L—length of the neutral line;

R—inner bend radius;

S is the sheet metal thickness;

K—position coefficient of the neutral layer;

α is the bend angle.

The K coefficient defines the position of the neutral layer:

$$K=t/S, \text{ where}$$

t is the distance from the inner bend surface to the neutral layer;

2.4.1.4.2. Definition of Flat Pattern Length by Setting the Bend Value

Flat pattern length **BA** of the cylindrical portion of a bend (short for *B end Allowance*) is defined by the user. The full flat pattern length **L** of a bend when unbending (see Fig.2.4.9) is calculated by the formula:

$$L = A + BA + B.$$



Fig. 2.4.9. Unbending a bend: a) bend is bent, b) bend is unbent

2.4.1.4.3. Definition of Flat Pattern Length by Setting Bend Reduction

The flat pattern length of the cylindrical part of the bend l is calculated by the formula:

$$l = 2 \cdot a - BD, \text{ where}$$

BD (abbreviation of *Bend Deduction*) — bend deduction; is set by the user,

a — geometrical bend parameter; is defined automatically by the system (Table). 2.4.1).

Tab. 2.4.1. Defining the parameter **a** for different bend angles α

$\alpha < 90^\circ$	$\alpha \geq 90^\circ$
$a = (R + S) \cdot \text{tg}(\alpha/2)$	$a = (R + S)$

The full flat pattern length **L** of a bend when unbending (see Fig.2.4.10) is calculated by the formula:

$$L = A' + B' - BD.$$



Fig. 2.4.10. Unbending a bend: a) bend is bent, b) bend is unbent



For bend angles greater than 90 degrees, the bend deduction value **BD** can be negative.

2.4.1.4.4. Tables of Bends

While defining flat pattern length using the above-mentioned methods, values of the flat pattern parameter (**K**, **BA** or **BD**) can be set randomly, taking into account the existing restrictions.

Dependency of the flat pattern parameter on the material thickness, angle and bend radius can be found, for example, experimentally and put together into tables. In the KOMPAS system, those are called **bend tables** and can be used for defining flat pattern lengths.

A table type is determined depending on its contents:

- ▼ Table of position coefficients of the neutral layer
- ▼ Table of bend amounts;
- ▼ Table of bend deductions.

Tables of bends are stored in files with extension *loa*.

KOMPAS-3D Distribution includes four bend tables:

- ▼ Two tables of position coefficients of the neutral layer:
 - ▼ *Sample1_K_factor.loa*,
 - ▼ *Sample4_K_factor.loa*,
- ▼ Table of bend amounts *Sample2_Bend_allowance.loa*;
- ▼ Table of bend deductions *Sample3_Bend_deduction.loa*.



The described tables contain sample data and are provided as an example. Use them as a template to create your own tables of bends.

Examples of bend tables are located in the folder defined by the KOMPAS-3D system variable *SHEETMETAL*. By default, this folder is *..Sys\SHEETMETAL* relative to the main system installation folder.

Using Tables of Bends

To be able to use a specific bend table in sheet bodies, you need to set this table file first in the sheet body properties dialog (see Section 9.2.7.4.8 on p. 1851) or using command **Operations— Sheet body elements— Table of bends— Select**.

To get the flat pattern parameter for the bends of a new or edited sheet feature of the current part from the table, complete the following steps:

1. On the **Options** tab of the Property Bar, access the **Method of Defining Flat Pattern Length** list.
2. Select a **Bend Table** row from the list.

In the field **Source File** you will see the name of the bend table selected when setting up sheet body properties.

If the **Bend Table** option was enabled when setting up sheet body properties, then, to define flat pattern length for all new bends the default method will be **Bend Table**. In that case, you do not need to perform the described steps.

After entering the values for the angle and bend radius from the bend table, the flat pattern parameter will be set that corresponds to those values. It will be used when unbending a created sheet feature.

When creating the first sheet feature in a model, which uses a bend table, all data from the table is written in the model. No relation is established between the model and the table file.

Any bends, whose flat pattern parameters come from a table, will use the data stored in the document. The source table file can be changed or deleted. That will not affect bend parameters.



Only one bend table can be stored in a model.

Change of the Table of Bends

To use another table file for bends created by a table, you would need to change the bend table stored in the model. For this, do any of the following:

- ▼ Call the command **Operations — Features of Sheet Body — Table of Bends — Select**.
- ▼ Click the **Browse** button in the sheet body properties dialog for the current model (see Section 9.2.7.4.8 on p. 1851).



Changing the bend table we will cause a change in the properties of all sheet body features, whose flat pattern parameter was obtained from the table: this parameter value will be selected from a new table. The icons of modified features will be marked by red “checks” in the Design Tree, signifying that rebuilding is required.

Table of bends stored in a model can be recorded into a file. To do this, call command **Operations — Sheet body elements — Table of bends — Save as...** In the appearing standard Windows dialog specify the filename to write the table to.

Table of bends can be deleted from the model. To delete the bend table from a part, call the command **Operations — Features of Sheet Body — Bend Table — Delete**. In this case, the method of defining the flat pattern length will change automatically for the sheet features created using a table. It will correspond to the type of the used bend table. The flat pattern parameter value will stay equal to the tabulated value for the current radius, bend angle and material thickness.

For example, a feature was created using a table of bend amounts. After deleting the table, this element's **Bend amount** will be defined by the method of defining flat pattern length. The bend amount will be still equal to that previously selected from the table.

A sheet body may contain **features with bend angles determined automatically**. Such elements include:

- ▼ Sheet body with an open sketch (bend angles are defined by the sketched object parameters)
- ▼ Incisions with the preset size less than the minimal one (bend angles are calculated using the formula, see Table 2.4.9 on p. 281).

If these elements use a bend table, then, upon deleting the latter, the elements are processed as follows.

The method of defining the flat pattern length is automatically replaced by the method corresponding to the used table type. The flat pattern parameter takes the **default value** (see Section 9.2.7.4.8 on p. 1851).

If the above-mentioned sheet elements contain bends with switched on definition of parameters based on the source object (i.e., option **Based on the source item** is active, see Section 2.4.4.3.6 on p. 260), the screen displays the bend selection dialog (Fig. 2.4.11).

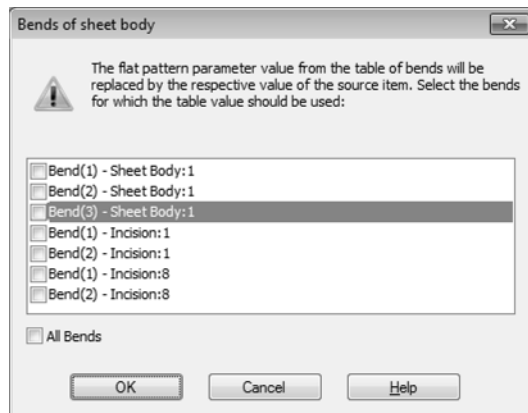


Fig. 2.4.11. Bend Selection dialog

It contains the list of bends having defined properties, which allows setting up processing of those bends upon the table deletion.

In the bend selection dialog, mark the bends, for which you want to preserve the flat pattern parameter obtained from the table. Click **OK**.

For selected bends, the parameter definition by the source object will be automatically disabled. The method of defining the flat pattern length is replaced by the method corresponding to the used table type. The flat pattern parameter of each bend will assume a value selected from the table according to the angle value of that bend.

For other bends, the property of parameters being defined by the source object will be preserved. Therefore, each of them will receive the same parameters as the source object, regardless of the bend angle.

Format of table of bends

A Bend Table is a text file with the *.Joa* extension. Table files can be viewed and edited using any text editor, such as Windows Notepad. Following is an excerpt from a table of position coefficients of the neutral layer.

Table Type — the table of K coefficients

1

Interpolation — apply linear interpolation

1

<0.5> 1.0 2.0 3.0

0.0 0.375 0.415 0.439

360.0 0.375 0.415 0.439

<1.0> 1.0 2.0 3.0

0.0 0.350 0.375 0.398

360.0 0.350 0.375 0.398

Bend tables are created according to the following rules.

- ▼ The measurement units are not entered in the table. Linear values to measure it in millimeters, angular in degrees.
- ▼ Lines starting with the “#” character are comments.
- ▼ The table may contain empty lines. Those are ignored when processing the table.
- ▼ The first value in the table defines its type:
 - 1 — the table of **K** coefficients;
 - 2 — the table of bend amounts **BA**;
 - 3 — the table of bend reductions **BD**.
- ▼ The second value governs use of interpolation for finding intermediate parameter values:
 - 0 — do not apply interpolation;
 - 1 — apply linear interpolation.



If interpolation is allowed then the table is used as follows.

- ▼ If the specified values for the sheet body thickness, radius and bend angle exactly match a table entry, then the flat pattern parameter is obtained from the table.
- ▼ If the specified value of the thickness, radius or bend angle does not match any entry in the table, then the flat pattern parameter is calculated using the linear interpolation.

If interpolation is not used and the specified value of the sheet body thickness, radius or bend angle does not match any entry in the table, then the bend cannot be created. The respective sheet feature is marked in the Design Tree as failing. To fix the failure, you need to bring the bend parameters to a match with the table or select another bend table.

- ▼ Bend Table consists of several **blocks of data**. Each block contains the values for the parameter **K**, **BA** or **BD** for various thicknesses, radii and bend angles. The block structure is the same.

- ▼ The sheet material thickness is constant throughout a data block. Its value is entered in brackets. For example, in the above table fragment, the thickness value is in the upper-left corner of the block.
- ▼ The first line of the block contains bend radius values. Those are positioned in the ascending order left to right.
- ▼ The left column of the block contains bend angle values. Those are positioned in the ascending order top down.
- ▼ Blocks in a table are positioned in the order of increasing material thickness.
- ▼ Values in a table are separated by spaces or tabs.



If a table is used, then the values are not extrapolated: if the thickness, bend radius or angle is outside the range of table arguments, then an error message is displayed.

2.4.1.5. Chamfer

While building such sheet elements as bends (see Section 2.4.4 on p. 252) and stamp elements (see Section 2.4.8 on p. 311), the user can switch on or off display of the phantom on the screen in his discretion.



To manage display of the phantom for the current (created or edited) sheet element, button **Display element phantom** is used in the Special Control Panel.

The ghost of a sheet feature is the wireframe image of the feature with the current parameters. Upon changing settings, the ghost dynamically updates.

If no ghost is displayed (while the **Show Element Ghost** button is pressed) means that the sheet feature cannot be created with current settings.

Using ghost makes a model creation more convenient, since that supports a visual inspection of the creation result. There are cases, however, (for example, when creating beats) when a sheet feature is rather complex, and its ghost drawing (or updating) is slow. In such cases, we recommend switching off the ghost display to speed up work.

2.4.2. Sheet Body

It is created by extruding a sketch in the direction perpendicular to its plane.

Before creating a sheet body, you need to create a sketch in the model, which will define the body shape.

2.4.2.1. Sketch Requirements

Requirements to a closed sketch of a sheet body:

- ▼ Sketch objects can be a part of one or several contours,
- ▼ If there are several contours, one of them must be the outer one and the rest must be nested in it.
- ▼ only one level of nesting is permitted for contours.

Requirements to an open sketch of a sheet body:

- ▼ sketch objects should comprise one contour,
- ▼ a sketch may consist of lines and circle's arcs only,
- ▼ arcs must be connected with lines and other arcs in tangent points.

2.4.2.2. Creating Sheet Body



To create a sheet body in a model, call the **Sheet body** command.



The command **Sheet Body** is available, if at least one sketch is selected.

Set sheet body parameters in the tab of the Properties panel **Parameters**.

If there several bodies in a model, in addition to parameter setup setting of operation application area can be necessary. For this, the **Operation Result** tab of the Properties panel is used. Detailed information on adding material operations application area set it is contained in Section 2.3.10.3.1.



Having finished setup, confirm performance of the operation.



In Window model a sheet body will be displayed with the preset parameters, and the Design Tree will display an image of a sheet body.

After creation of a sheet body, the model forms a certain set of variables (for more information see Section 2.4.1.3 on p. 233).

2.4.2.2.1. With the closed sketch

If the sketch is closed, it is necessary to set its following parameters to build a sheet body.



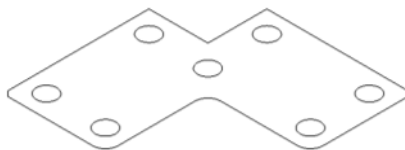
- ▼ Direction extrusion of the sketch. The forward direction is shown by the arrow in the Model window. For the Direction option group serves to switch the **direction**.



- ▼ Sheet body thickness. In fact, the thickness is the distance of extruding the sketch.



- ▼ Definition of length of the flat pattern of the bend (see Section 2.4.4.3.5 on p. 259). If the flat pattern length is taken from the table of bends, you can change the default table by pressing button **Select another table of bends**.



a)



b)

Fig. 2.4.12. Building of a sheet body based on the open sketch: a) sketch, b) sheet body

2.4.2.2.2. With the open sketch

The following is special about creating a sheet body based on an open sketch:

- ▼ Line segments in the sketch make the flat portions of the sheet body,
- ▼ Arcs in the sketch make bends of the respective radii,
- ▼ Angles in the sketch a contour make bends with the user-specified inner radius.

Select direction and depth of extrusion. This is done in the same way as when building extrusion features (see Sections 2.3.2.4.2 on p. 156 and 2.3.2.5 on p. 157).



Select direction to add material: outside or inside relative to the surface, formed by moving the sketch in the indicated direction.

Enter the thickness of the material layer to be added (sheet body thickness) in the **Thickness** field.

Enter the inner **radius for bends** corresponding to corners in the contour, in the Bend radius field (Fig. 2.4.13).



The zero bend radius is not allowed. The minimum value is *0.0002mm*.

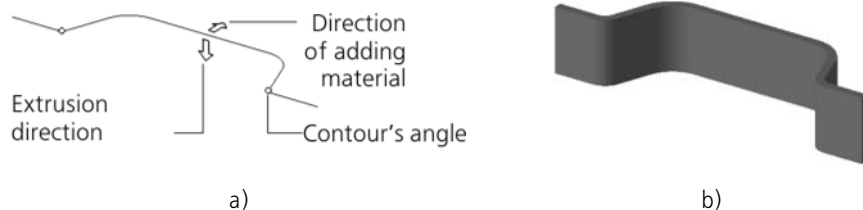


Fig. 2.4.13. Building of a sheet body based on the open sketch:
a) sketch, b) sheet body



Set up the definition of the flat pattern length of the bend (see Section 2.4.4.3.5 on p. 259). If the flat pattern length is taken from the table of bends, you can change the default table by pressing button **Select another table of bends**.

The option **Unbend** controls the state of a sheet body. If it is enabled then construction will result in unbend sheet body. When the option is switched on, all the bends of the sheet body are unfolded.



Icon an unfolded sheet body is marked in the Design Tree by the sign “unfolded”. Management of the attribute “unfolded” for a sheet body is performed in the same way, as for other sheet elements that contain bends (see Section 2.4.4.3.4 on p. 258)

Bends made as a result of building of a sheet body with the open sketch, are not different from bends made using special commands. Bends of a sheet body are displayed in the Design Tree in the same way, as other bends (see Section 2.4.4.2 on p. 253), any bend of a sheet body can be edited (see Section 2.4.4.3.6 on p. 260).

2.4.3. Shell

The shell is formed by moving the sketch in the direction perpendicular to its plane and by adding thickness of the resulting surface (Fig.2.4.14).

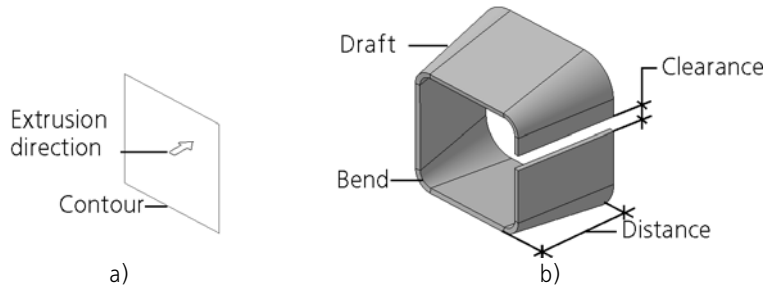


Fig. 2.4.14. Creating a shell based on the closed contour:
a) sketch, b) shell

The sketch may be both closed and open contour. If the contour is closed, the shell will automatically be cut by its height. Gap value and position can be specified by the user. There are several methods of cutting the shell's edges.

Bends with specified radius are automatically constructed in the shell's contour corners.

You can perform the same operations with respect to the shell as to a sheet body: unbending/bending, creation of cuts and holes, incisions, beats, and so on.



The shell sections that correspond to curved portions of the sketch that are not circle arcs can be unbent as ordinary bends. At the same time, such sections result not from bending operations, but from operations of deforming; the result of their unbending is conditional to a certain extent and does not depend on the parameter that defines the flat pattern length and value of the parameter.

2.4.3.1. Requirements to the sketch of the shell

- ▼ Sketch objects should comprise one contour. No smooth mate between contour objects is required.
- ▼ A contour can be closed or open.
- ▼ The contour shall not have self-intersections.

2.4.3.2. Shell construction

Before creating a shell you need to create a sketch in the model, which will define its shape.



To create a shell in the model, call the **Shell** command.

Complete the following steps.

1. Select the sketch.



Sketch can be specified prior to calling the command.

2. Set the following parameters:

- ▼ **shell parameters**, see Section 2.4.3.2.1,
- ▼ **base weld parameters** (see Section 2.4.3.2.2),
- ▼ **parameters of the joint's welds and gap** (for a shell with a closed sketch), see Section 2.4.3.2.3,
- ▼ **gap position** (for a shell with a closed sketch), see Section 2.4.3.2.4.

If there several bodies in a model, in addition to parameter setup setting of operation application area can be necessary. For this, the **Operation Result** tab of the Properties panel is used. Detailed information on adding material operations application area set it is contained in Section 2.3.10.3.1.

All parameter values, once entered or modified, are immediately displayed on the screen as the ghost of the extruded feature. A ghost allows a visual control over parameter accuracy.



To set such parameters as direction and distance of the construction, wall thickness, slope angle, bend radius, gap width and position, you can use hot points (for more information about hot points, see section 2.1.4.2.2 on p. 138).



3. Click **Create Object** button to complete the construction.

Construction result— new body, pasted or cut element— appears in the graphic area.



The shell's icon appears in the Design Tree.

When creating a shell, the model forms the same set of variables as while creating a sheet body (for details see Section 2.4.1.3 on p. 233).

2.4.3.2.1. Shell parameters

To create a shell, it is necessary to set up the following parameters.



- ▼ **Direction of plotting.**

The forward direction is shown by the arrow in the Model window. **Direction** switch group serves to change the direction.



- ▼ **Plotting distance.**

To select the method for specifying the distance, list **Method** is used. The following methods are available: **At Distance**, **To Vertex** and **Up to Surface**. Parameters are defined in the same way as when definition of the extrusion depth (see Sections 2.3.2.5.1 on p. 157, 2.3.2.5.3 on p. 158 and 2.3.2.5.4 on p. 159).



- ▼ **Draft angle.**

To define the slope direction use the **Draft angle** switch group, to define the angle of a slope — **Angle** field.





▼ Shell thickness.

To change the direction of adding the material, the switch group **Direction for thickness** is used, while to add the width value — the **Thickness** field.



▼ Bend Radius.

Bends of the specified radius are built in the joints of the contour segments.

Depending on the selected thickness direction, a bend will be created, whose outer or inner radius is equal to the specified value.

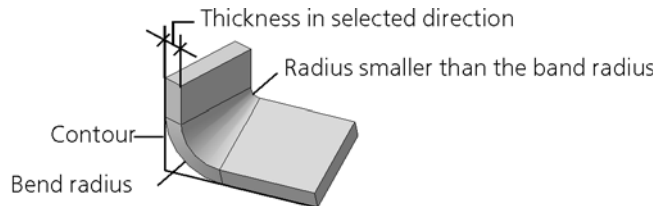


Fig. 2.4.15. Bend radius of shell with a slope

Possible reduction of the shell with a slope radius should be taken into account. Construction is performed if the current thickness radius (lower than the bend radius) has an admissible value. The minimum radius value is *0.0002 mm*.

- ▼ **Flat Pattern Parameters** (see Section 2.4.1.4 on p. 237). These parameters can only be applied to shells with the contour in sketch containing segments and/or arcs (if the contour includes other objects, flat pattern parameters do not affect unbending of bends formed by these objects).

2.4.3.2.2. Base Weld parameters

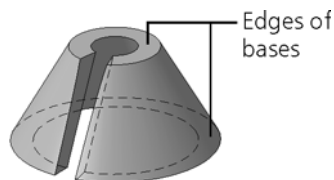


Fig. 2.4.16. Shell's base welds'

Parameters of the shell's base welds' are specified in the **Weld** tab of the Property bar.

Select a type of the shell's base welds' form in the group **Base welds**.



If the **Perpendicular to the sheet's plane** switch is on, the welds are cut perpendicular to the shell's flat pattern plane (Fig.2.4.17, a).



If the **Matching base surfaces** switch is on, the welds are cut parallel to the sketch plane (Fig.2.4.17, b).

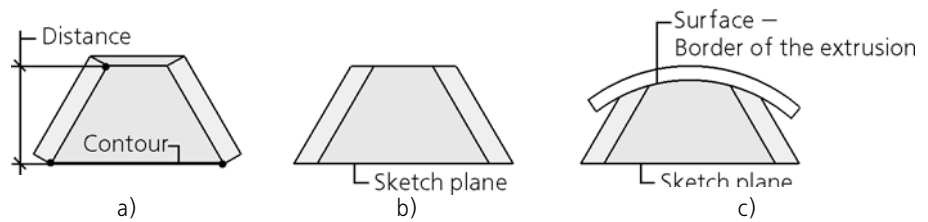


Fig. 2.4.17. Base welds on the conical shell: a) Perpendicular to the sheet's plane; b) Matching base surfaces (by At Distance method); c) Matching base surfaces (by Up to Surface method)

The system constructs base welds as follows.

Sketch contour is extruded at the distance, specified in the **Method** group. Thickness is added to the resulting surface— inside or outside. The welds are cut by the surfaces, perpendicular to the shell's surface, or by parallel surfaces, one of which coincides with the sketch plane.

If the extrusion method **To surface** is selected, the **Matching base surfaces** method will mean cutting one edge by the sketch's plane and another— by the surface (Fig.2.4.17, c).

2.4.3.2.3. Parameters of the joint's welds and gap

For shells based on the closed contour, you need to specify parameters of the joint's welds and gap value.

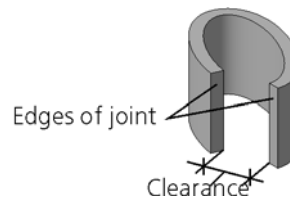


Fig. 2.4.18. Joint's welds of the shell

Enter or set by meter the gap size (distance between edges) in the **Gap** field in the **Edges** tab of the Property Bar. A zero value cannot be set.

In group **Joint's welds** select an option of joint's welds.



When selecting the method **Perpendicular to the sheet's plane** switch is on, the welds are cut by the surfaces, perpendicular to the shell's¹ surface (Fig. 2.4.19, a).



When selecting the method **Parallel to each other**, the welds are cut by parallel surfaces in the direction of the gap (Fig. 2.4.19, b).

1. The edge's surface is constructed so that it is perpendicular to the shell's surface along the whole gap. Edges of a non-flat shell with gap slope are linear surfaces, while in other cases the shells' edges are flat.

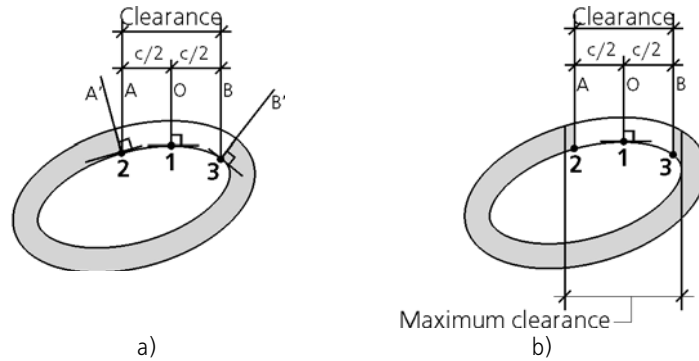


Fig. 2.4.19. Joint's welds on the shell:
a) Perpendicular to the sheet's plane; b) parallel to each other



The maximal acceptable value of the gap depends on its location on the shell— the gap should not protrude beyond the internal wall in the direction which is perpendicular to the gap direction (see Fig.2.4.19, b). Gap position on a shell can be changed, see Section 2.4.3.2.4 on p. 251.



If a contour is not closed, then the welds are always cut perpendicular to the sheet's plane.

Weld construction and gap measuring

When the shell has no slope, the system constructs joint's welds as follows.

1. In the plane parallel to the sketch's plane and crossing the shell, intersection lines are built for the plane and the shell (Fig.2.4.19). As a result, two equidistant curves are obtained.
2. In the **t1** gap's central point, the **O** plane is constructed perpendicular to the curve's tangent, as is shown on the figures.
3. At distances equal to half of the gap's value, which are indicated as $c/2$ on the figures, two planes **A** and **B** parallel to it are constructed. The planes intersect the curve in points **p2** and **p3**.
4. The edges' surfaces are built:
 - ▼ When selecting the method **Parallel to each other**, the welds are cut by parallel surfaces **A** and **B** (Fig. 2.4.19, b),
 - ▼ when selecting the **Perpendicular to the sheet's surface** method, **A** and **B** planes are constructed perpendicular to tangents in **t2** and **t3** points (Fig.2.4.19, a), while the edges are cut by these planes.



To create **t2** or **t3** points, the system selects the curve with the least curve radius on the gap's sections (Fig.2.4.20).

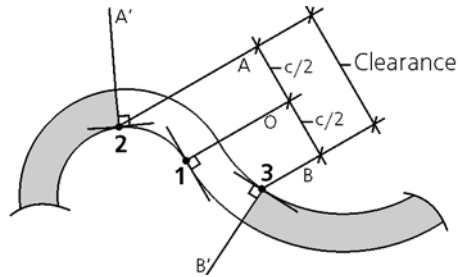


Fig. 2.4.20. Aside from the joint, when placing the gap on the shell's bend without a slope section by **Perpendicular to the sheet's plane** method

If the shell has a slope, joint's welds are created as in the case of the shell without slope, except for the case when gap is created on a non-flat part using the **Perpendicular to sheet surface** method. In this case, the edge is a linear surface whose generating line is perpendicular to the external and internal surfaces of the shell (Fig.2.4.21).

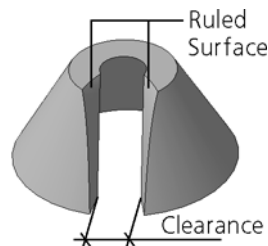


Fig. 2.4.21. Joint's welds when placing the gap on the conical shell by **Perpendicular to the sheet's plane** method

The gap is measured in the shell projection on the plane, parallel to the sketch plane.

The gap value is equal to the minimal distance between the edges in the direction of the tangent to the curve in the gap's point (Fig.2.4.20 and 2.4.21).

2.4.3.2.4. Gap Displacement

The displacement is set from the initial position of the gap— a point in its center— to the same point in the new position. In default position of the gap, its offset is zero.

To set the displacement, select the method in the **Gap displacement** group of the **Edges** tab of the Property Bar, and then enter the value for the displacement.



▼ **By Parameter U** — by the relative offset from the initial to the new position of the gap. The **U Parameter, %** field is used to set the percentage value of the contour length in the sketch.



▼ **By the length** — by the length of the contour from the initial to the new position of the gap. The **Length** field is used to specify the absolute offset in the length measurement units.



▼ **Angle of the position** — by the central angle from the initial to the new position of the gap. Variant is available if shell contour is a circle. The **Angle** field is used to set the central angle of the arc in angle measurement units.

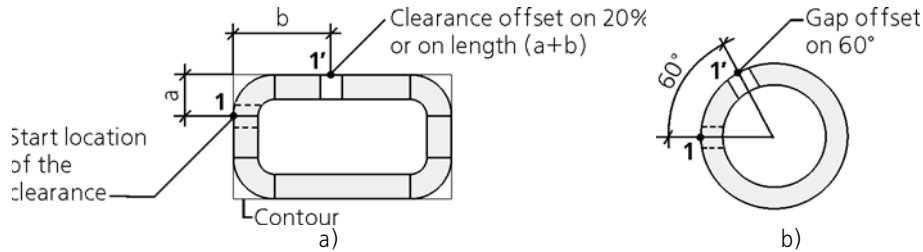


Fig. 2.4.22. Gap displacement:
a) by parameter U or by the length; b) by the central angle

Enter or specify by counter the value of the offset parameter.

When moving the point by mouse, the parameters are calculated and put to these fields.

During construction, you can switch between the options. In this case the offset value will be calculated and displayed in the units of the respective selected option.

2.4.4. Bends

Bend is the cylindrical portion of a sheet body (Fig. 2.4.23).

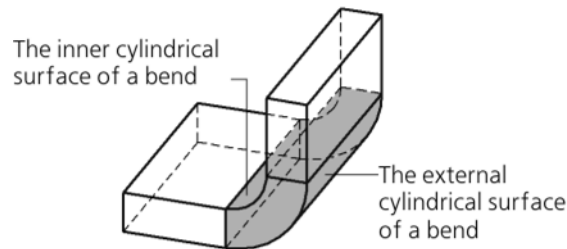


Fig. 2.4.23. Example of a bend in a sheet body

Bends can be formed in a model by two methods:

1. while creating a sheet body based on the open sketch (see Section 2.4.2.2.2 on p. 244);
2. by using the commands



▼ **Bend,**



▼ **Bend by Sketch,**



▼ **Bend by Line,**



▼ **Release.**

This section is devoted to bends received by the second method. Subsection 2.4.4.3 contains the description of general methods to create these bends. Subsections 2.4.4.4, 2.4.4.5, 2.4.4.6 and 2.4.4.8 describe additional methods that can be used when working with commands **Bend**, **Bend by line**, **Incision** and **Bend by Sketch**, respectively.



Bend creation, as well as corner capping commands can not be applied, if a sheet body comprises several parts (for example, if a body is divided by the cut operation). To create bends, modify a body to make it single-piece by editing existing elements or creating new elements joining all parts.

2.4.4.1. Terms and definitions

To create a bend, it is necessary to have the **bend line** — a linear object that defines position of the bend in a sheet body.

A flat external or internal face of a sheet body that contains the bend line, is called the **basic face** of this bend.

If the part is bend in the direction towards the base face, then the bend direction is considered **direct**. When bending towards the opposite side, the bend is considered to be in the **reverse direction**.

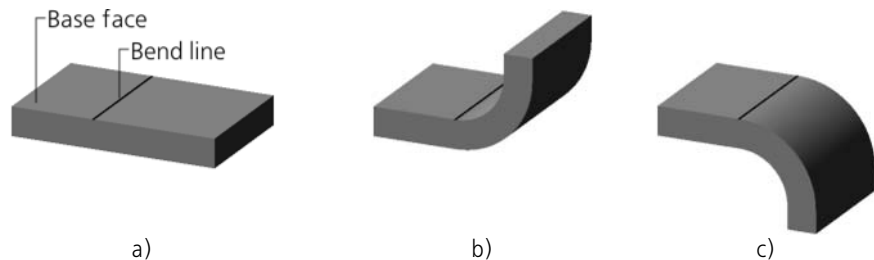


Fig. 2.4.24. Bend direction: a) basic face and bend line, b) forward-direction bend, c) back-direction bend

2.4.4.2. Display of bends in the model's Design Tree

Icons of bends are located in the Design Tree in “branches” of their **source objects**, those sheet elements that contain them (Fig. 2.4.25).



Fig. 2.4.25.

Some elements, for example, an incision, may refer to several bends. To view the list of bends, expand the “branch” of the Design Tree, corresponding to the original sheet feature.

Each bend is automatically assigned a name. The name contains the ID number of the bend in the sheet feature and that feature name.

The context menu of the bend is practically similar to context menu of other sheet elements.

Using it, you can set up properties of a source element, and also view object relations and attributes.

The **Edit element** command starts up the editing process for the selected bend (see Section 2.4.4.3.6 on p. 260).



Exclude from **Calculation/Include into Calculation** affect the entire feature containing the selected bend, while the commands **Hide/Show** act on the entire part body.

2.4.4.3. General Methods to Build Bends

After calling any of the commands to create bends, the Properties panel displays control elements that allow to set up various parameters of bends.

Initially, these parameters (angle, radius, etc.) coincide with the default ones for the given model (see Section 2.4.1.3 on p. 233). If those are left unchanged, then default variables will be assigned to the parameters. In the future, this will help quickly changing parameters of same-type bends. You can find more information on working with variables in the sheet body in Section 2.4.1.3.



To set numerical parameters for bends, you can use typical points (see Section 2.1.4.2.2).



To fix the created bend, press button **Create object** in the Special Control Panel.

A new feature will be added to the sheet body being developed, and the respective icon will appear in the Design Tree:



▼ Bend



▼ sketch bend,



▼ bend by line,



▼ incision.

You can create several components of the same type (bends, bends by line or incisions) without exiting the command. To create the next element select its source object (or objects); if necessary modify parameters of the operation and click the **Create Object** button.



If you need to build a few elements in sequence with the same parameters, perform the following operations. Click the **Object Auto Create** button on the Advanced Control Panel after you have created first object with correct parameters. Then select source objects for new elements. They will be formed automatically, i.e. without waiting for confirmation of creation.



To complete building of elements, press button **Interrupt command** in the Special Control Panel.

2.4.4.3.1. Direction of count and interpretation of the angle value

When creating a bend, the angle can be counted in either direction from the base face. The **forward** (“direct”) **direction** of counting the angle is outward with respect to the part’s body. This direction is defined by the system automatically and is indicated by a ghost arrow in the Model window.



To select the angle direction — **Forward** or **Reverse** — activate the appropriate switch in the **Direction** group.



The value entered in the **Angle** field can be interpreted as follows

▼ **Bend angle** or

▼ **Appending Angle.**





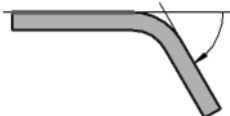
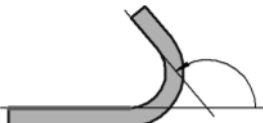


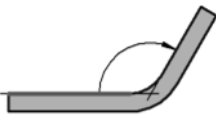
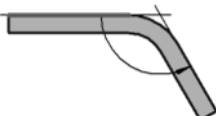
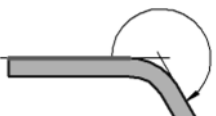



To select interpretation of the angle value, activate the right switch in group **Angle sense**.



Direction of bend, direct or back one (see Fig.2.4.24, b, c) depends both on the direction of angle count and its interpretation (see Table). 2.4.2).

Tab. 2.4.2. The scheme of creating a bend depending on the angle counting direction and sense*

	 Forward angle counting direction	 Reverse angle counting direction
 Bend angle		
		
 Complimentary angle		
		

* The scheme shows the part projection (mapping) on the plane perpendicular to the bend line. The base face projection is shown as a thickened line.



You cannot create a bend with the complimentary angle equal to 180° , since that means no bending: Bend angle is equal 0° .

If the angle sense is the bend angle, then the angle counting direction is the same as the direction of the resulting bend.

If, on the other hand, the angle sense is complimentary, then these directions will call in side only with the angle values being within the range from 0° to 180° . When the complimentary angle values are in the range from 180° to 360° , then the direction of the resulting bend is opposite to the selected direction of counting the angle.

E.g. Figure 2.4.26 the bend is displayed in the **back** direction. The angle of the bend is equal to 250° , it has the **complimentary** sense and is applied in the **opposite** direction.

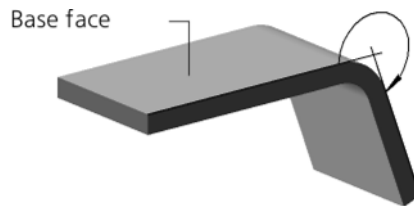


Fig. 2.4.26. Bend in the back direction

2.4.4.3.2. Radius of the Bend



To select the method to set radius, activate the right switch — **Internal radius** or **External radius** — in group **Method to set radius**.



Depending on the chosen method of defining the radius, a bend will be created, whose outer or inner radius is equal to the specified value.

Minimal value of internal radius — $2 \cdot 10^{-4}$ mm. The inner and outer radii are related as follows:

$$R_{\text{outer}} = R_{\text{inner}} + S,$$

where S is the material's thickness.

Therefore, the minimum value of the outer radius is equal to the sum $(S + 2 \cdot 10^{-4})$ mm.

2.4.4.3.3. Angle Release

If the created bend is located relative to the adjacent bend (bends), for instance, as shown in Fig. 2.4.27, you can use the release of angle.

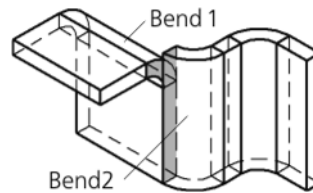

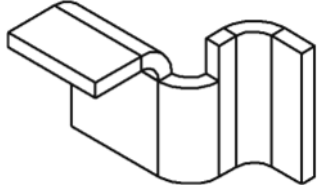

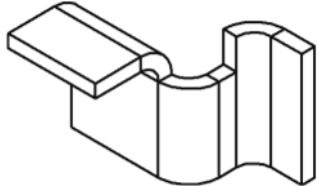

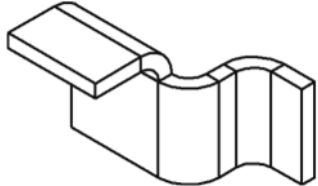


Fig. 2.4.27. Example of location of bends, that allows to use release angle: the flat ring face of the created bend1 partially coincides with the flat rectangular face of the adjacent Bend 2

Three release methods are available (Table 2.4.3). To select the right method, activate the corresponding switch in the tab **Release** of the Properties panel.

Tab. 2.4.3. Methods to release bend angles

	The value of option Release of angle	Construction result*
	Bend proper	
	Bend and its extension	
	All bends	

* This column presents the results of use of various methods to release an angle when building Bend 1 (see Fig.2.4.27).

Use of the release angle in the **Bend** command is subject to the following specifics.

1. Release of the angle is possible only if the line of the bend being created is perpendicular to the lines of the adjacent bends.



2. To enable the release, you need to activate the '**Release angle**' switch on the tab **Release** of the Property Bar. After that, the release type switches will become available.

2.4.4.3.4. States of bends

A bend can be created either in the bent or in the unbent state. The state of a sheet element is managed by option **Unfold**. If it is switched off, the result of building will be a sheet element with folded bends (Fig.2.4.28, b). If the option is switched on, bends of a sheet element will be unfolded (Fig. 2.4.28, c).

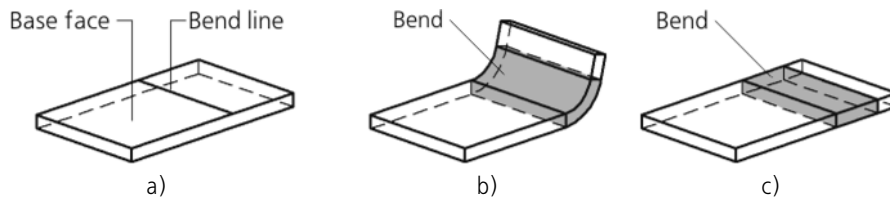


Fig. 2.4.28. Building of a bend: a) basic face and bend line, b) folded bend, c) unfolded bend

The unbent sheet feature icon is marked by the “unbend” glyph in the Design Tree.



- ▼ unbent bend,
- ▼ unbend bend by sketch,
- ▼ unbent bend by line,
- ▼ unbent incision.

The attribute “unbent” can be managed both using option **Unbend** (while creating or editing a sheet element) and commands **Unbend** and **Bend** in the context menu of a sheet element in the Design Tree. Upon calling one of them for some element, the other command appears in its context menu. When multiple elements are selected, both commands are available (Fig. 2.4.29).

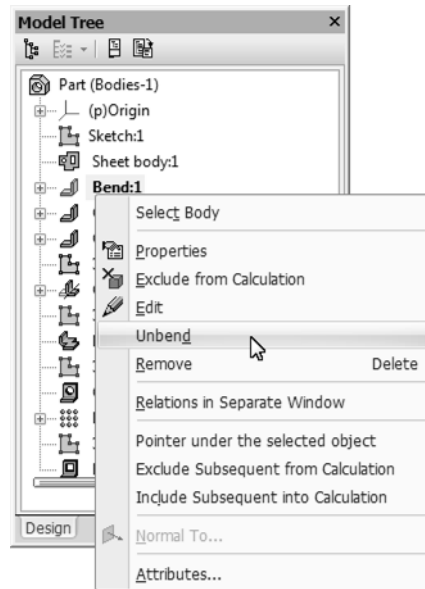


Fig. 2.4.29. Commands **Bend** and **Unbend** in the Bend Contextual Menu



There also exist special commands to edit a sheet body, that allow to change the state of one or several bends (see Section 2.4.7.1 on p. 302).

2.4.4.3.5. Setup of definition of flat pattern length

To set the method to define length of the flat pattern of the bend, open the list **Flat Pattern** and select the right line in it:

- ▼ **Coefficient,**
- ▼ **Bend amount,**
- ▼ **Reduce bend,**
- ▼ **Table of bends.**

The procedure to define flat pattern length for each of the methods is considered in detail in Section 2.4.1.4 on p. 237.

Depending on the chosen method, a field will appear on the Property Bar for entering the value of the flat pattern parameter (**Coefficient**, or **Amount**, or **Reduction**) or a field for selecting a bend table file.

By default, the bend table file is used that was selected in the sheet body properties dialog for the current model. Selection of another table of bends for the current model is described in Section *Change of the Table of Bends* on p. 240.

2.4.4.3.6. Editing of Bend Parameters

By default, bend parameters correspond to those set during setup of a sheet body. Part of parameters for a specific bend can be changed. To edit a bend, select it in the Design Tree and call the command **Edit Feature** from the context menu or from the **Editor** menu.

The Properties panel will display elements that allow to set up the bend (Fig. 2.4.30). By default, option **By Source Item** is switched on. This means that bend parameters should be the same, as for the source object.

To define individual bend parameters, enable the option **By Source Item**. After that, other control elements will become available.

You can change the following bend parameters:

- ▼ Method of defining radius,
- ▼ Bend radius value,
- ▼ Method to define flat pattern length and its parameters.

Other bend properties correspond to those of the source object.

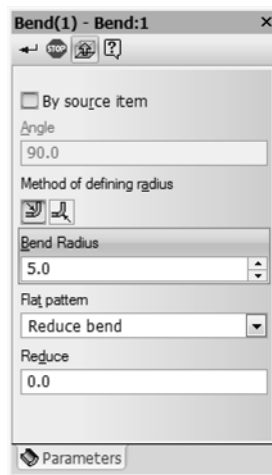


Fig. 2.4.30.



You can neither alter the method of defining the radius nor change its value when editing bends of a sheet body with an open sketch, corresponding to arcs in the sketch of the sheet body.

Bends with disabled parameter definition by the source object are displayed in the Variables window as standalone elements dependent on their source object (see Section 2.4.1.3.1 on p. 235, item5).

2.4.4.4. Bend

You can create a bend along the edge of the sheet body. The edge must be straight and belong to an outer or inner flat face of the sheet part.

The indicated edge will be considered the bend line, and the face that contains it (external or internal) as the basic face of the bend (Fig. 2.4.31).

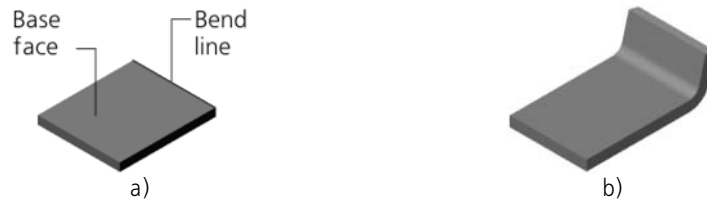


Fig. 2.4.31. A bend along the edge: a) basic face and bend line, b) result of building



To add to a sheet body a bend along the edge, call **Bend** command.

Indicate the edge, along which the bend is to be located.

Select the direction of count and angle interpretation (see Section 2.4.4.3.1 on p. 254).

Indicate the value and the method of defining bend radius (see Section 2.4.4.3.2 on p. 256).

Set up release of angle (see Section 2.4.4.3.3 on p. 256).

Select the bend state (see Section 2.4.4.3.4 on p. 258).


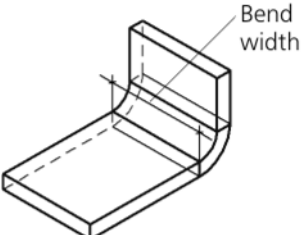
Set up the definition of the flat pattern length of the bend (see Section 2.4.4.3.5 on p. 259).

The phantom arrow (that indicates the forward direction of angle count) is located at the **left end** of the edge. The bend side that is closer to the left end of the edge, is also considered as **the left one**, while the opposite bend side— as **the right one**. Left and right sides are to be distinguished when setting up the location of the bend and parameters of its lateral sides.


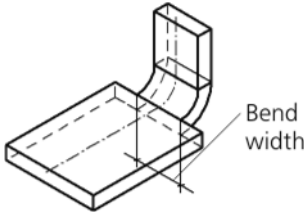

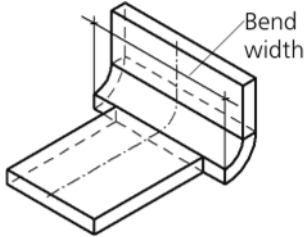

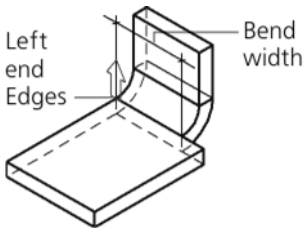
2.4.4.4.1. Bend Placement

When creating a bend, there are different choices for positioning it on an edge. To specify the bend position, select the desired option from the **Location** list (see Table 2.4.4). Depending on your choice, the fields appear in the Property Bar for entering the bend width or its offsets from the edge ends.


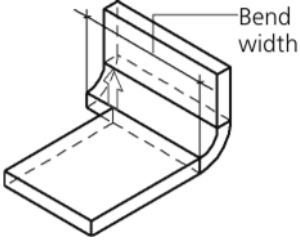

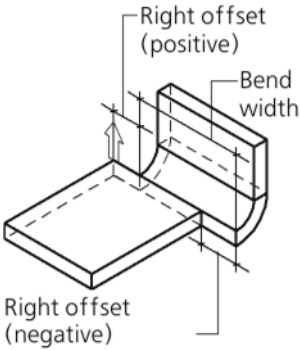
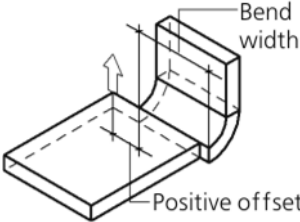
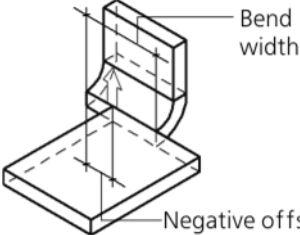
Tab. 2.4.4. Bend location options

	The value of option Location	Rules to locate the bend and define its width	Construction result
	Through all length	The width of the bend is equal to the edge length.	


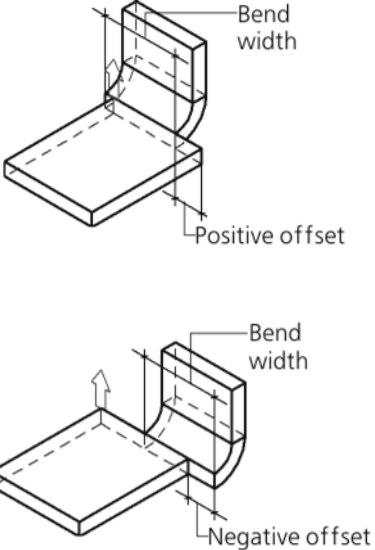
Tab. 2.4.4. Bend location options

The value of option Location	Rules to locate the bend and define its width	Construction result
	<p>Centered</p> <p>The bend width is entered in the Width* field. The bend is positioned centered by the edge midpoint.</p>	
	<p>Left</p> <p>The width value is entered in the Width* field. The bend is positioned so that the left end of the edge lies in the plane of the left side of the bend.</p>	
	<p>Right</p> <p>The width value is entered in the Width* field. The bend is positioned so that the right end of the edge lies in the plane of the right side of the bend.</p>	

Tab. 2.4.4. Bend location options

The value of option Location	Rules to locate the bend and define its width	Construction result	
	Two offsets	<p>The bend width is automatically defined as the difference between the edge length and the total offset amount^{**}. The right and left offset values are entered in the fields Offset 1 and Offset 2, respectively. The bend position is determined by the offset dimensions.</p>	
	Offset at left	<p>The bend width and the offset amount are entered in the fields Offset^{***} and Width[*]. The bend is positioned so that the distance from the left end of the edge to the left side of the bend is equal to the specified offset value.</p>	
			
			

Tab. 2.4.4. Bend location options

The value of option Location	Rules to locate the bend and define its width	Construction result
	<p>Offset at right The bend width and the offset amount are entered in the fields Offset^{***} and Width[*]. The bend is positioned so that the distance from the right end of the edge to the right side of the bend is equal to the specified offset value.</p>	

* The bend width can have any positive values. Zero width is not allowed.

** Indent values can be both positive and negative. When calculating the bend width, indent symbols are taken into account: positive indents are laid off inside relative to the part body, and negative ones— outside. Indent values which make the bend width negative or equal to zero, and also values which separate the bend from the part body, are not allowed.

*** An offset value can be either positive or negative. A positive offset is made inward with respect to the parts body, the negative— outward. Offset values that make the bend detached from the part's body are not allowed.

2.4.4.4.2. Bend Extension

A **bend extension** is a portion of the sheet body adjacent to the bend on the side opposite to the edge, along which this bend is positioned (Fig 2.4.32).

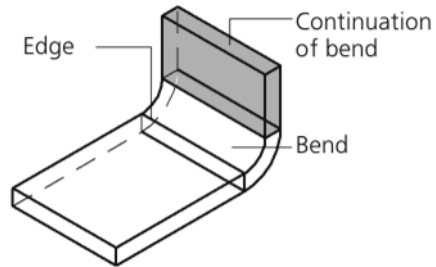

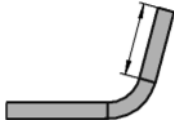
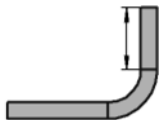
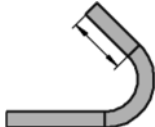



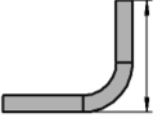
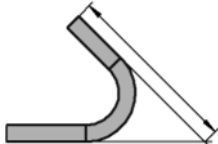
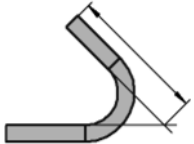


Fig. 2.4.32. Bend extension along the edge

Enter length of bend extension in field **Length**.


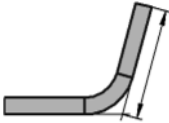
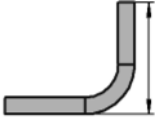
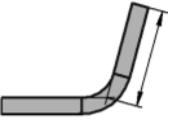
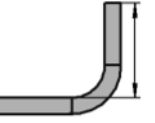
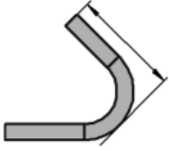
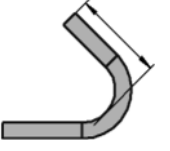
Indicate the method to set length for bend extension, selecting the right option from list **Method to set length** (Table 2.4.5). When using the methods **Length by contour** and **Length by Tangency**, the **Inside** option is available, that allows to select a surface limiting the extension length.

Tab. 2.4.5. Chart of defining length of bend extension*

Option value	Inside option is off	Inside option is on
Method to set length		
 Length		
		
 Length Along Contour		
		

Option **Inside** is not available when using method **Length**.

Tab. 2.4.5. Chart of defining length of bend extension*

Option value	Inside option is off	Inside option is on
Method to set length		
 Length by tangency		
		
		

* The scheme shows the part projection (mapping) on the plane perpendicular to the bend line.

Peculiarities to Set Parameters for Bend Extension

- ▼ When using the **Length** method, it is possible to enter the zero length. This means the bend will be created without an extension (Fig. 2.4.33).

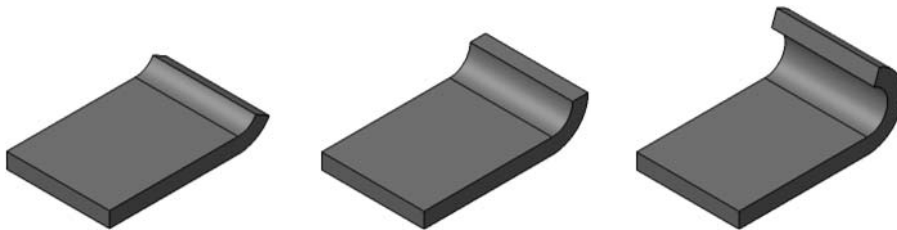


Fig. 2.4.33. Examples of bends without extension

- ▼ When using the methods **Length by contour** and **Length by tangency**, the value entered in the **Length** field must be greater than a certain minimum value (Fig. 2.4.34). In a general case, it depends on the material thickness, angle and bend radius.



Fig. 2.4.34. Minimal length

- ▼ Defining **Length by contour** is possible for angles from 0° to 180°.

2.4.4.4.3. Bend Offset

Bend shift is a parameter characterizing the bend offset in the base face plane. The bend shift is made perpendicular to the edge, along which it is positioned.

To indicate the type of shift of a bend relative to the edge, select the right option from list **Shift Type**.



With shift when offsetting a bend inward or outward, the offset distance is defined by the user arbitrarily. The **Offset** field is provided to enter this distance. With zero shift the bend starts directly from the edge (Fig. 2.4.35, a), and with positive one— it moves inside or outside (Fig.2.4.35 b, c)

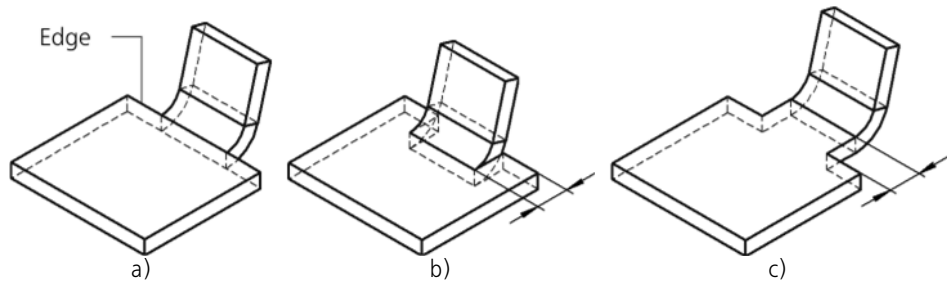





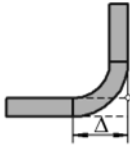
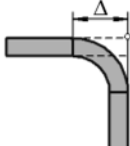
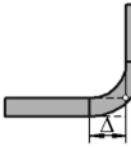
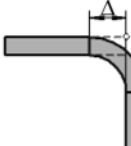
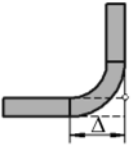
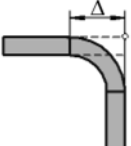
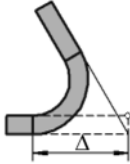
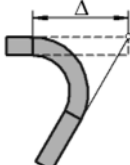
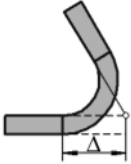
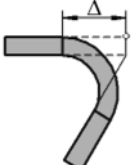
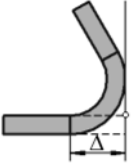
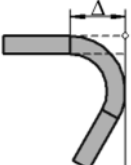
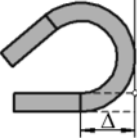
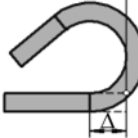
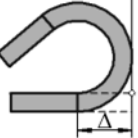
Fig. 2.4.35. Bend Offset a) zero, b) inward, c) outward

For other offset types, the offset distance is automatically determined by the system. This distance depends on the bend angle, material thickness and the inner radius of the bend (Table 2.4.6).




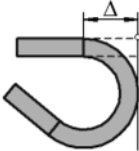
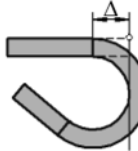
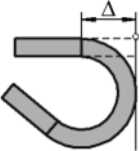
Tab. 2.4.6. Building chart and shift value*

Bend angle α	By external contour line	By internal contour line	Tangent to bend
from 0° to 90°			
	$\Delta = (r + s) \cdot \text{tg}(\alpha/2)$	$\Delta = r \cdot \text{tg}(\alpha/2)$	$\Delta = (r + s) \cdot \text{tg}(180 - \alpha/2)$

Tab. 2.4.6. Building chart and shift value*

Bend angle α	By external contour line	By internal contour line	Tangent to bend
			
90°	  $\Delta = r + s$	  $\Delta = r$	  $\Delta = r + s$
from 90° to 180°	  $\Delta = (r + s) \cdot \text{tg}(180 - \alpha/2)$	  $\Delta = r \cdot \text{tg}(180 - \alpha/2)$	  $\Delta = r + s$
more than 180°			

Tab. 2.4.6. Building chart and shift value*

Bend angle α	By external contour line	By internal contour line	Tangent to bend
			
			
	$\Delta = r + s$	$\Delta = r$	$\Delta = r + s$

* The chart indicates the model projection to a plane perpendicular to the edge, along which the bend is located. The edge projection is designated with a circle.

2.4.4.4.4. Lateral Sides

When creating a bend along the edge, two methods to manage parameters of lateral sides of the bend are possible:

- ▼ Change of width of bend extension (Fig. 2.4.36)

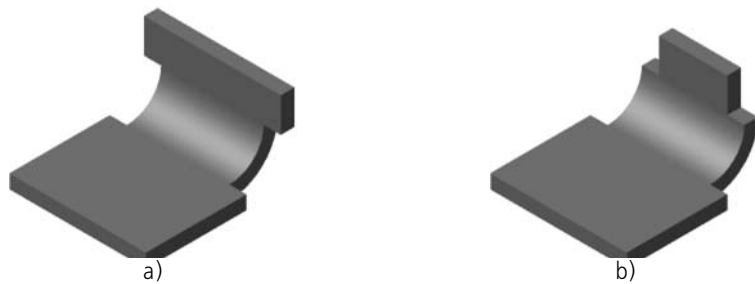


Fig. 2.4.36. Change in the bend extension width: a) increase, b) reduction

- ▼ Defining the bend angle and/or the slope of lateral sides on the bend extension (Fig. 2.4.37).

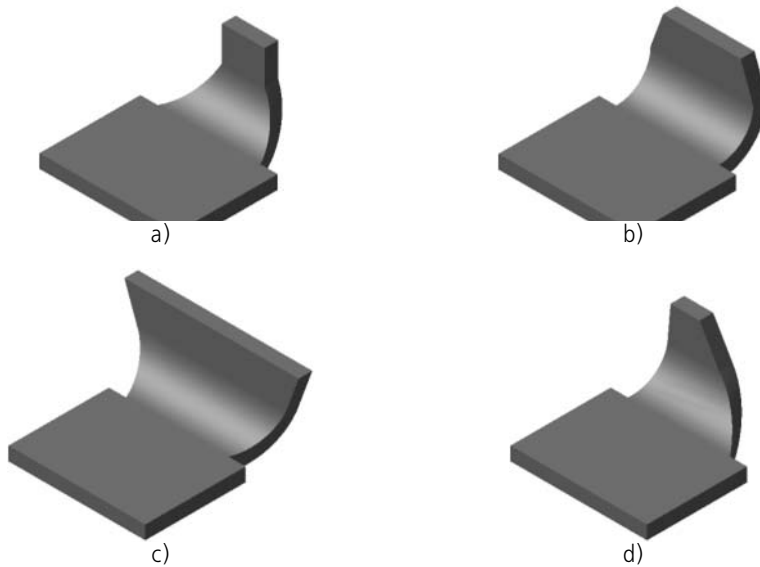


Fig. 2.4.37. Managing end sides of the bend
 a) angles at the bend, b) the slope of side faces of the extension,
 c), d) angles at the bend and the slope of side faces of the extension



To select a setup option for the left lateral side, activate the right switch **Slope and Angle** or **Extension** in group **To the Left**.

To select a setup option for the right lateral side, activate the right switch **Slope and Angle** or **Extension** in group **To the Right**.

Slope and Angle

Rules of defining the angle of a side wall slope and the angle at the bend:

- ▼ Angles are measured **on the unbent bend**.
- ▼ Angles are counted from the plane perpendicular to the edge, along which the bend is positioned.
- ▼ Angles can be in the range from -90° to $+90^\circ$.
- ▼ Positive angles are counted inward with respect to the bend body, the negative— outward.

To enter the inclination angle of the bend extension side wall use the **Slope** field.

To enter the bend angle, use the **Bend Angle** field.



If the bend angle is a nonzero, then the respective site face of the bend will not be planar.

Extension

Widening of bend extension is the change in the width of the bend extension as compared to the bend width.

The widening value can be negative, positive or zero. A negative widening means a reduction in the width of the bend extension, positive is an increase in the width of the bend extension, zero— no change in the width.

Use the **Extension** field to enter the value of the bend extension widening.

2.4.4.4.5. Bend Release

Release of bend is made by slots in the sheet part made on the sides of a bend.

Release can have a rectangular or round shape. Dimensions of release are defined by its depth and width (Fig. 2.4.38).

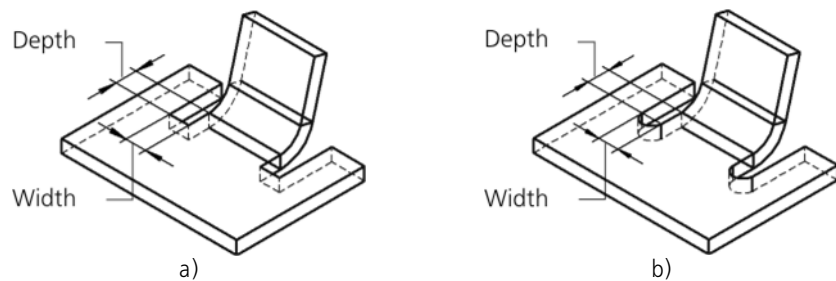


Fig. 2.4.38. Bend release: a) rectangular, b) rounded



To form release of bend, activate switch **Release of Bend** in the tab **Release** of the Properties panel.



To select shape of release, activate the right switch— **Rectangular** or **Round**— in group **Type**.



Enter the dimensions of the release in the fields **Depth** and **Width**. Values of depth and width can be zero or positive.

The option **Account for Bend Width** controls placement of releases with respect to a bend. If release widths are to be included into the bend width (in this case, the actual bend width decreases, see Fig. 2.4.39, b), activate this option. When the option is switched off, release widths are not included into the bend width (the bend width doesn't change, see Fig. 2.4.39, c).

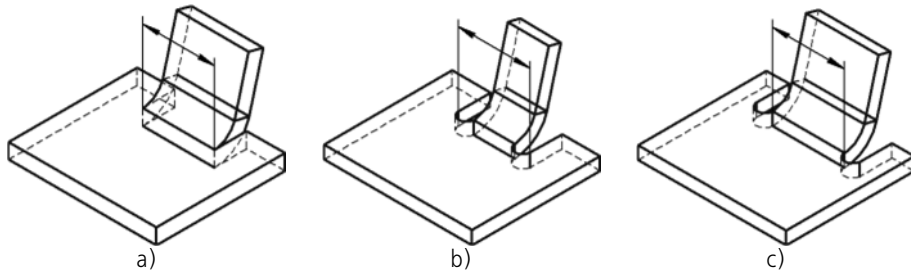


Fig. 2.4.39. Location of releases: a) the bend width without releases, b) account for bend width in releases, c) don't account for bend width in releases

If the angle at the bend (see Section 2.4.4.4 on p. 269) is not equal to zero, release of bend rotates (Fig. 2.4.40). Rotation angle of release is equal to angle at the bend. In this case, the release depth is counted along the side face of the unbent bend, while the release width is perpendicular to that face (Fig. 2.4.41).

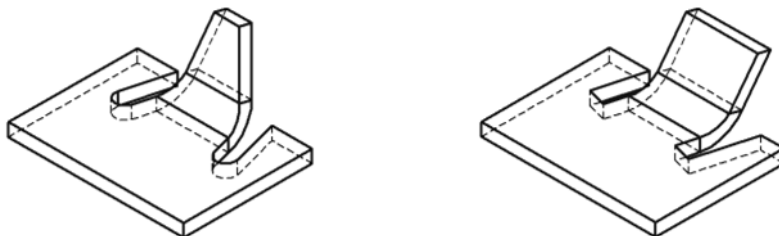


Fig. 2.4.40. Release of bend with angle at the bend not equal to zero

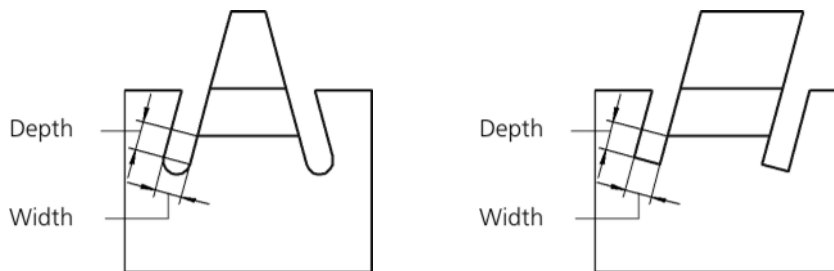


Fig. 2.4.41. Definition of dimensions for release of bend with angle at the bend not equal to zero

2.4.4.5. Bend by Line

You can bend a sheet body by straight line relative to some external or internal flat face of this part. The selected line and face will be considered the bend line and the base face of the bend.



To bend a sheet body by line, call command **Bend by Line**.

Indicate the basic face and the bend line (see Section 2.4.4.5.1).

Select the direction of count and angle interpretation (see Section 2.4.4.3.1 on p. 254).

- Indicate the value and the method of defining bend radius (see Section 2.4.4.3.2 on p. 256).
- Set up release of angle (see Section 2.4.4.3.3 on p. 256).
- Select the bend state (see Section 2.4.4.3.4 on p. 258).
- Set up the definition of the flat pattern length of the bend (see Section 2.4.4.3.5 on p. 259).

2.4.4.5.1. Original objects and construction result

The bend line should have the following parameters:

- ▼ Line segment of a sketch, constructed on outer or inner flat face of a sheet body,
- ▼ Straight edge of a geometry-creating feature or surface,
- ▼ polyline segment,
- ▼ Auxiliary axis.

Bend line requirements:

- ▼ The bend line must lie in the plane of the base face,
- ▼ The bend line should not coincide with the edge of the basic face.
- ▼ The bend line should cross or have a common point at least with one edge of the basic face (in the last case, extension of the bend line should cross the contour of the basic face, i.e., the common point should not be the contact point for the edge and the bend line).
- ▼ The bend line can be fully located inside the contour of the basic face (in this case, one or both end points of the bend line can be located at edges of the face).



The auxiliary axis is infinite. Therefore, when selecting an axis to be the bend line, it doesn't matter, if the section depicting this axis crosses the contour of the basic face or not (see Fig. 2.4.44).

The result of bend creation depends on the relative position of the base face and the bend line. A general rule is: the bend is performed for the part section that includes the basic face or a section of the basic face that crosses the bend line or its extension.



If a straight line segment or a polyline segment is used as a bend line, and the face being bent has a complex shape, then, for the bend to be constructed correctly, we recommend positioning the bend line in such a way as to explicitly go through all portions that are meant to be bent.

Fig. 2.4.42– 2.4.44 indicate options of bends for a sheet body, built using command **Bend by Line**. The upper flat face is used as the basic face in all the given cases; the bend line is displayed as thickened.

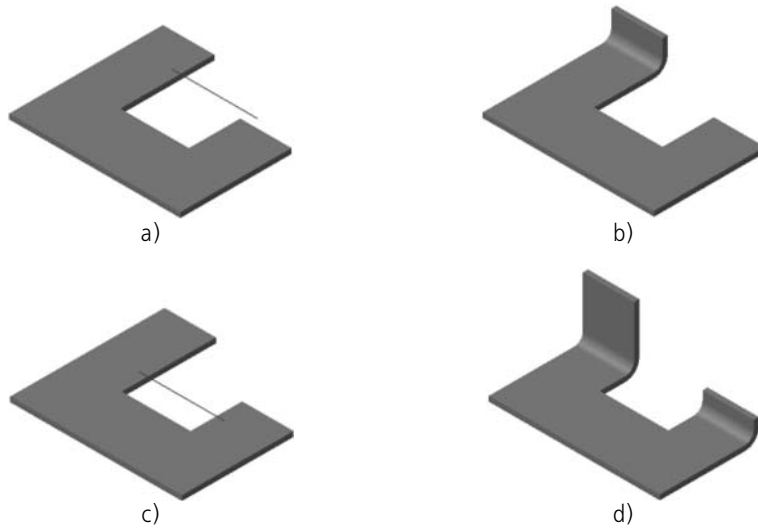


Fig. 2.4.42. Dependency of the resulting bend on the bend line position: a), c) options of position for the bend line, b), d) corresponding results of building



Fig. 2.4.43. Bending a part with a hole: a) position of the bend line, b) the result of building



Fig. 2.4.44. Using an auxiliary axis as the bend line:
a) location of the axis, b) the result of building

2.4.4.5.2. The Fixed Side of the Bend

When building a bend by line, the model window displays the phantom arrow (Fig. 2.4.45). It indicates the forward direction of count of the bend angle and its **fixed side**, the section of the basic face, the position of which will not change during the bend.

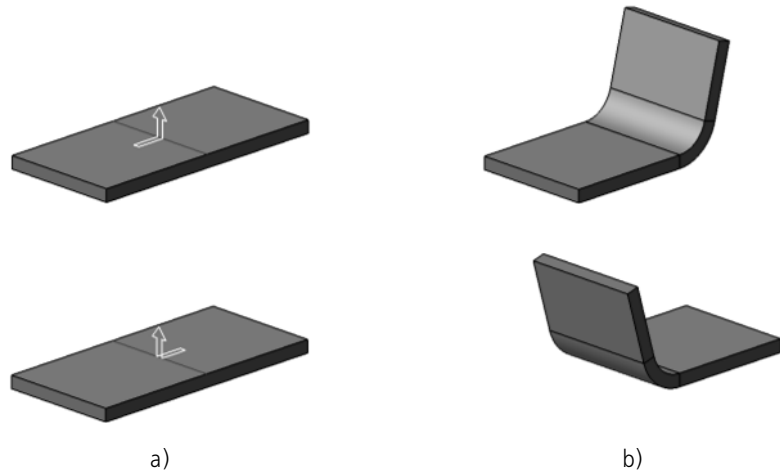


Fig. 2.4.45. Bend by line: a) direction and fixed side, b) the result of the operation



To make another side fixed, use the group of switches **Fixed Side**.


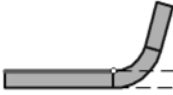
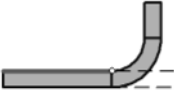
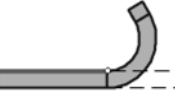





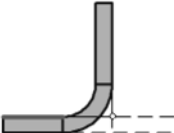
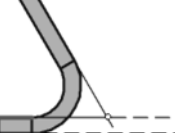


Change of the fixed side immediately affects the arrow position in the model window.


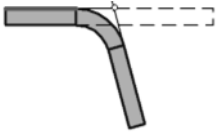
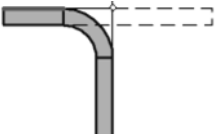
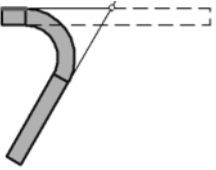

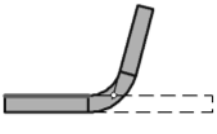
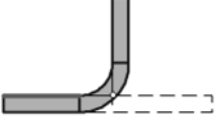
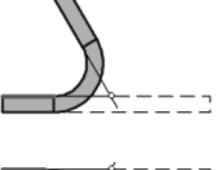
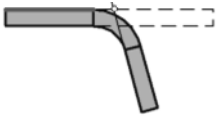
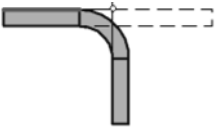
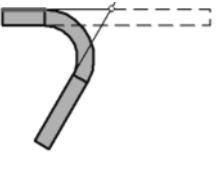
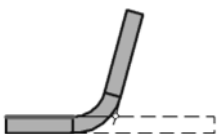
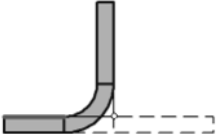
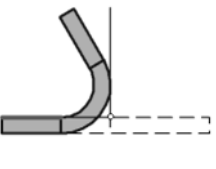
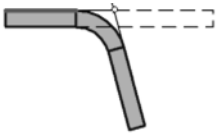
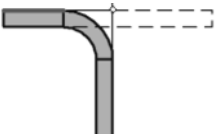
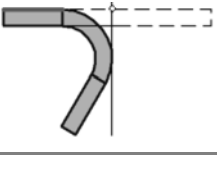
2.4.4.5.3. The Method to Form the Bend

When building a bend by line, you can select the method to form it from list **Bend Method** (Table 2.4.7). The table indicates the schemes to build bends for various values and directions of angle count.

Tab. 2.4.7. The scheme to build a bend, depending on the forming method*

Value of option	Scheme of creating		
Bend method			
 By Bend Line			
			
 Bend line outside			

Tab. 2.4.7. The scheme to build a bend, depending on the forming method*

Value of option Bend method	Scheme of creating			
	Bend line inside			
	Tangent			
				
				
				

* The scheme shows the part projection (mapping) on the plane perpendicular to the bend line. The base face protection is marked by a thickened line, and the bend line projection— by a circle.

2.4.4.6. Incision

Serves to create two straight-line bends at once in a part with respect to some face of the part. The selected line and face will be considered the bend line and the base face of the incision.



To build an incision, call command **Incision**.

Specify a base face of the release and a bend line (see Section 2.4.4.6.1).

Select the fixed side (see Section 2.4.4.5.2 on p. 274).

Select the direction of count and angle interpretation (see Section 2.4.4.3.1 on p. 254).



The value of the bend angle of the incision should be more than 0° and less than 180°; complementary angle— more than 0° and less than 360°.

Indicate the value and the method of defining bend radius (see Section 2.4.4.3.2 on p. 256).

Select the method to form the bend (see Section 2.4.4.5.3 on p. 275).

Set up release of angle (see Section 2.4.4.3.3 on p. 256).

Select the bend state (see Section 2.4.4.3.4 on p. 258).

Set up the definition of the flat pattern length of the bend (see Section 2.4.4.3.5 on p. 259).



The specified value and method of defining the radius, as well as the default method of defining flat pattern length — are used for both incision bends. Later on, you will be able to assign individual parameters to each of the bends (see Section 2.4.4.3.6 on p. 260).

2.4.4.6.1. Original objects and construction result

The following linear objects can be used as the bend line for incision:

- ▼ Line segment of a sketch, constructed on outer or inner flat face of a sheet body,
- ▼ polyline segment,
- ▼ auxiliary axis,
- ▼ Straight edge of a geometry-creating feature or surface.

Bend line requirements:

- ▼ The bend line must lie in the plane of the base face,
- ▼ The bend line must have at least one common point with the base face.



An auxiliary axis appears in the model as a straight line segment. Nevertheless, the axis is an infinite line. Therefore, when selecting an axis as the bend line, it doesn't matter whether the axis-denoting line segment has common end points with the base face. For a bend to be created correctly, it is sufficient that the axis projection (extension) has common points with the base face.

The result of incision creation depends on the relative position of the base face and the bend line. A general rule is: bending is done on the portion of the part including the base face or fraction thereof that fully or partially contains the bend line.



If a straight line segment or a polyline segment is used as a bend line, and the face being bent has a complex shape, then, for the bend to be constructed correctly, we recommend positioning the bend line in such a way as to explicitly go through all portions that are meant to be bent.

Fig. 2.4.46–2.4.49 indicate incision options. The upper flat face of the part is used as the basic face in all the given cases; the bend line is displayed as thickened.

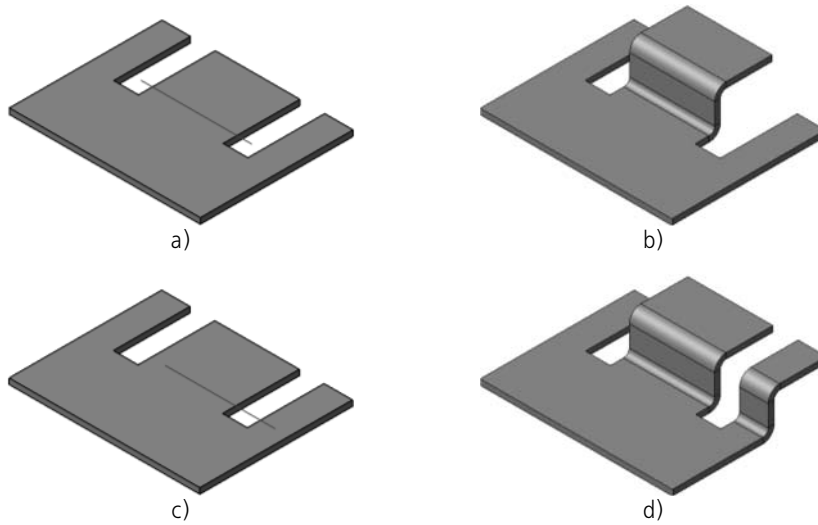


Fig. 2.4.46. Dependency of the construction result on the bend line position:
a), c) options for position of the bend line,
b), d) corresponding results of performance of the operation

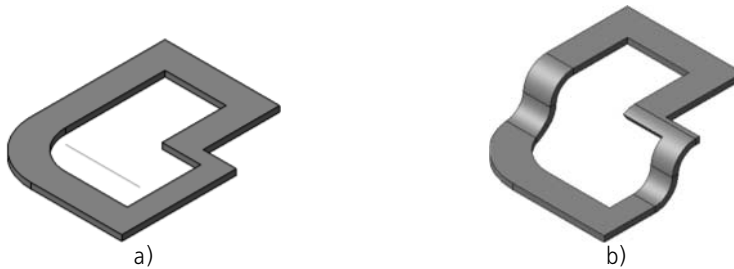


Fig. 2.4.47. Using an auxiliary axis as the bend line:
a) location of the axis, b) the result of building

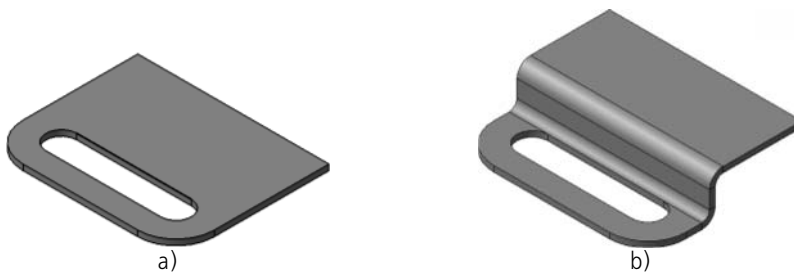


Fig. 2.4.48. The use of the edge as the bend line of incision:
a) location of the edge, b) the result of building

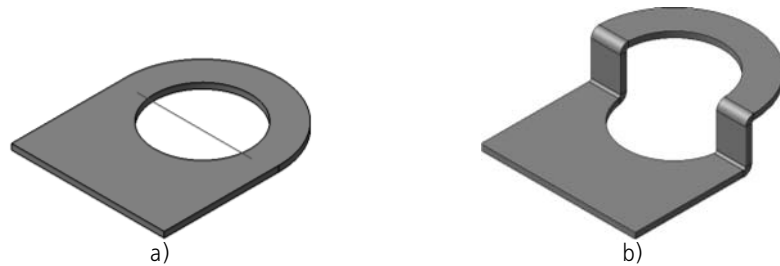


Fig. 2.4.49. Incision in a part of the hole:
a) position of the bend line, b) the result of building

2.4.4.6.2. Incision Size

Using switches of group **Dimension**, select the option to set incision height:



▼ **External dimension of incision,**



▼ **Internal dimension of incision,**



▼ **Full dimension of incision.**

Enter the value for dimension of incision in filed **Distance**.

2.4.4.6.3. Incision With and Without Adding Material

Option **With the Use of Material** allows to indicate if it is necessary to build incision with adding of material or without adding of material.

Incisions with and without adding material, created using the same and lines and base face, differ in the following properties:

- ▼ Shape of the rectangular incision projection on the base face
- ▼ unbent incision shape.

As an example, let's review a release introduced in a feature shown on Fig 2.4.50.

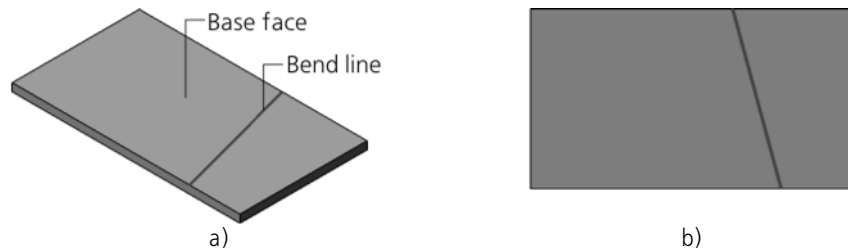
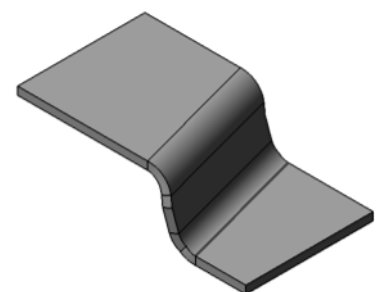
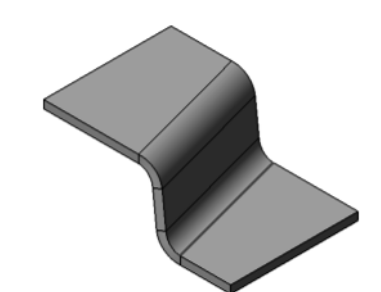

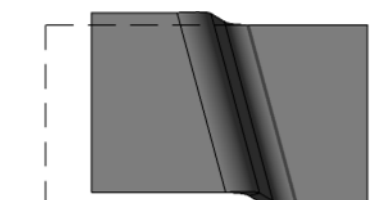
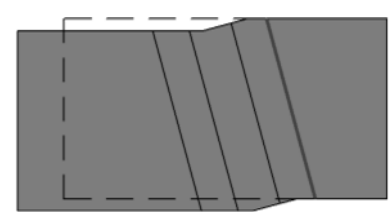
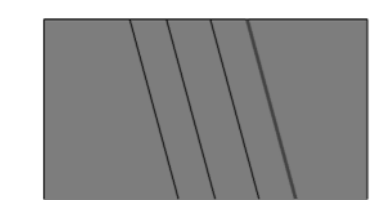


Fig. 2.4.50. The source incision element: a) isometric projection,
b) projection of the source element to the basic face

Comparison of results of incision building is given in Table 2.4.8.

Tab. 2.4.8. With and without adding material*

The differing properties are	Method of incision creation	
	With adding material	Without adding material
		
		
Incision projection on the base face	...coincides with mapped contour of the source element	...does not coincide with mapped contour of the source element
		
Unbent incision shape	...does not coincide with the shape of the source element	...coincides with the shape of the source element

* The bend line is shown as a thickened line, and projection of the source element— as a stroked one.

2.4.4.6.4. Flat Portion of the Incision

There can be a flat area between bends of the incision. Its presence depends on the proportion of preset (H) and minimal (H_{\min}) dimensions of incision (see Table 2.4.9).

H_{\min} — minimal internal dimension of incision without a flat area— is calculated using the formula:

$$H_{\min} = 2 \cdot R - (2 \cdot R + S) \cdot \cos \alpha, \text{ where}$$

R—internal radius of bends;
 S is the sheet metal thickness;
 α is the bend angle.

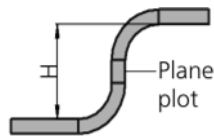
Tab. 2.4.9. Finding a flat portion in an incision*

Proportion between preset and minimal dimensions of the incision

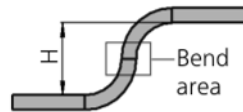
$H > H_{\min}$

$H > H_{\min}$

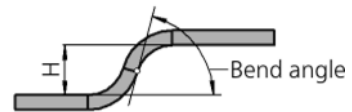
$H = H_{\min}$



Incision with a flat area. Values of H, R and α parameters correspond to the preset ones.



Incision without a flat area — with bend. Values of H, R and α parameters correspond to the preset ones.



Incision without a flat area. The H and R parameters have a higher priority, therefore the value entered in the **Angle** field will be ignored. The bend angle value is calculated by the system automatically based on the above formula.

* The figures show projection of incision to a plane perpendicular to the bend line of this incision.

2.4.4.7. Cap Corners

Capping of angle is a modification of two adjacent bends and their extensions (Fig. 2.4.51).

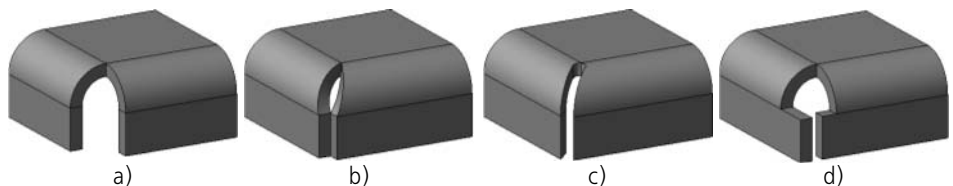


Fig. 2.4.51. Capping of angle: a) original part status, b), c), d) options of capping of angle

Adjacent are the bends that have a common edge positioned as shown on Fig.2.4.52.

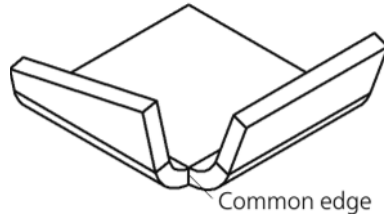


Fig. 2.4.52. Adjacent bends

Therefore, the bends with a release or those shifted with respect to each others are not considered adjacent (Fig. 2.4.53). Their respective corners cannot be capped.

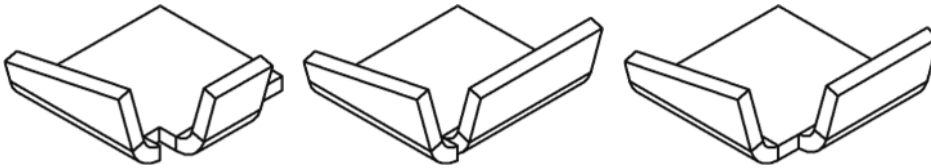


Fig. 2.4.53. Examples of non-adjacent bends

The flat portions of the part that are modified by capping the corner are called the **sides** of the corner.



Radii, angles and lengths of extensions of adjacent bends should be pairwise equal to cap corners accurately.

Capping of angle is characterized by the following parameters:


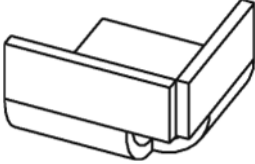

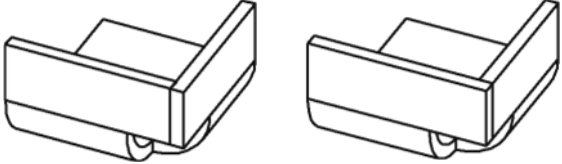

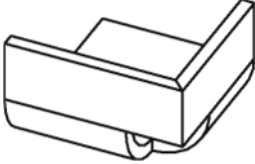
- ▼ Method (see Section 2.4.4.7.1),
- ▼ angle treatment (see Section 2.4.4.7.2),
- ▼ gap (see Section 2.4.4.7.3),
- ▼ extension (see Section 2.4.4.7.4).

The principle to perform capping of angle depends on the selected method. Building principles are described in Sections 2.4.4.7.5 and 2.4.4.7.6.

2.4.4.7.1. Methods of Capping

To indicate the capping method, open list **Type** on the tab **Parameters** of the Properties panel and select the right line from it (Table 2.4.10).

Tab. 2.4.10. Methods of capping of angle





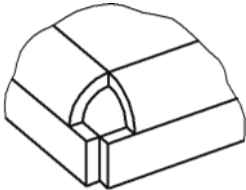
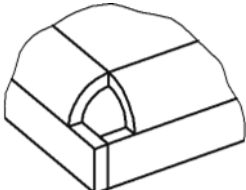
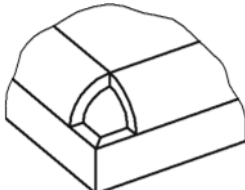
Value of option	Method	Construction result
	End-to-end closing	
	Capping with Overlap	
	Tight closing	

If you choose the **Capping with Overlap** option, a ghost arrow will be displayed in the Model window. It is located on the side of the angle that will overlap the second side. Click the **Change Overlapping** button to select another overlapping option. The arrow position will change in the Model Window.

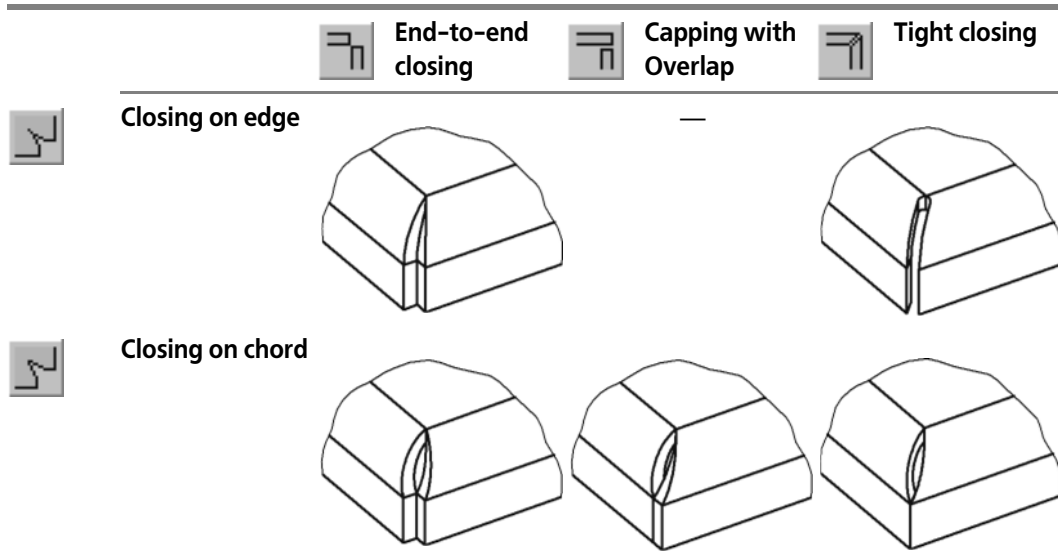
2.4.4.7.2. Angle Treatment

To indicate the angle treatment method while capping the angle, open list **Angle Treatment** and select the right line from it. Examples of corner treatment in connection with various capping methods are provided in the Table 2.4.11.

Tab. 2.4.11. Variants to process an angle using various methods during closing.

	 End-to-end closing	 Capping with Overlap	 Tight closing
 No ??processing			

Tab. 2.4.11. Variants to process an angle using various methods during closing.



Please note that the choices **Butt Juncture** and **Chord Juncture** are available if the adjacent bends have equal angles and equal radii. At the same time, the chord juncture is possible with all capping methods, whereas the butt juncture— only for the butt capping and tight capping.

If the **Tight Capping** is used with the **Butt Juncture** corner treatment, then the flat patterns of adjacent bends could overlap if the gap is zero or small. To avoid this, we recommend using such a gap which guarantees no overlapping.

2.4.4.7.3. Gap

Gap is the distance that will be maintained between two sides of a capped corner. The gap is measured in the part projection on the plane perpendicular to the corner sides. Such plane can be represented by any plane perpendicular to the intersection of outer or inner faces of the corner sides (Fig. 2.4.54, a). The **closable angle** is measured in the same plane (Fig. 2.4.54, b).

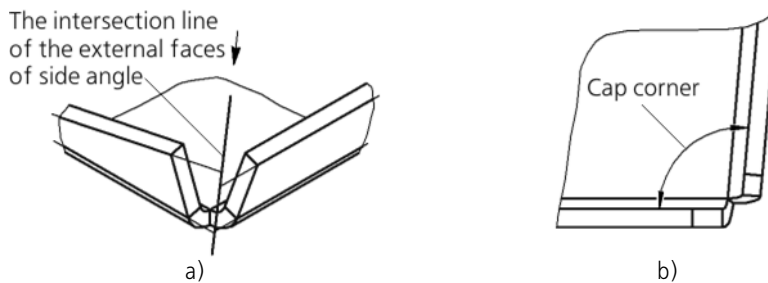


Fig. 2.4.54. Measurement of the gap: a) defining position of the measuring plane, b) part projection on the measuring plane (view by arrow); c) capping corners

To enter the gap value, field **Gap** in tab Parameters of the **Properties** panel is used.

2.4.4.7.4. Extension of Capping

Switch on option **Continue**, if it is required to cap matching bends, adjacent to sides of the closable angle.

To continue capping is possible if the following requirements are met:

- ▼ For adjacent bends:
 - ▼ Radii, angles and lengths of extensions of adjacent bends should be equal by pair.
- ▼ For abutting bends:
 - ▼ Bends have no offset and/or relief, and their radii and angles are equal by pair.
 - ▼ Bend placement on the edges is made in such a way that every bend begins from the edge vertex that is turned to the corner being capped.

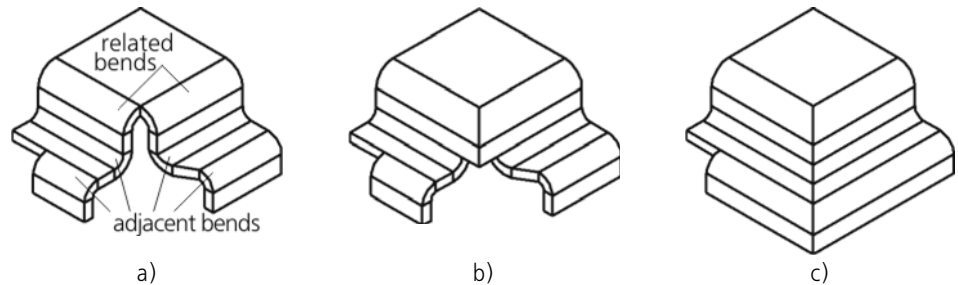


Fig. 2.4.55. Bend capping: a) original part status, b) capping without extension, c) capping with extension



Extension of capping is made up to the first pair of adjacent bends that do not meet the requirements.

2.4.4.7.5. The Principle to Build Line-On-Line Capping and Tight Capping

Let's consider the principle to build line-on-line capping and tight capping, using the part shown in Fig. 2.4.56 as an example.



Fig. 2.4.56. Butt capping: a) original part status, b) the result of capping of angle (the gap is not equal to zero)

The position of corner sides is determined as follows.

1. The bisector of the corner being capped is created in the plane in which the gap is measured (Fig. 2.4.57, a).
2. From each side of the bisector at the distance equal to half of the gap value, parallel lines are drawn (Fig. 2.4.57, a).
3. The corner sides are projected (extended) until reaching the obtained lines (Fig. 2.4.57, b). In tight capping, the corner sides are additionally modified: their lateral faces become parallel to the bisector (Fig. 2.4.57, c).

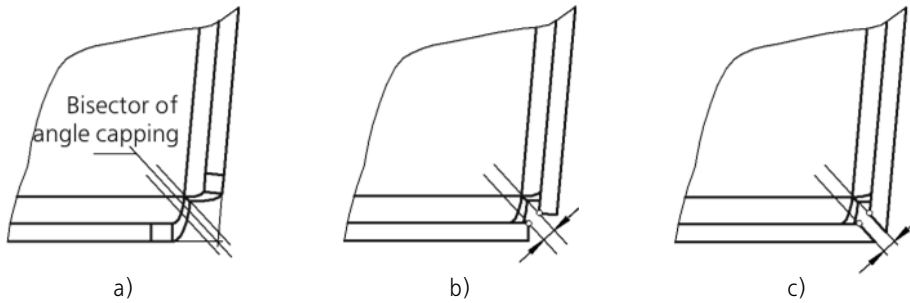


Fig. 2.4.57. Definition of position of angle sides with line-on-line capping and tight capping

The minimum gap amount is 0, the maximum is determined as shown in Fig. 2.4.58.

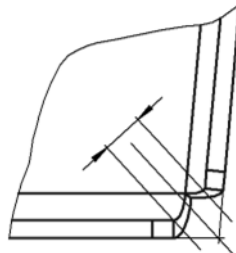


Fig. 2.4.58. In butt capping

2.4.4.7.6. The Principle to Build Capping with Overlap

Capping with overlap is constructed differently, depending on what is the corner to be capped— acute or obtuse.

Let's consider the principle to build capping with overlap, using parts shown in Fig.2.4.59 as an example.

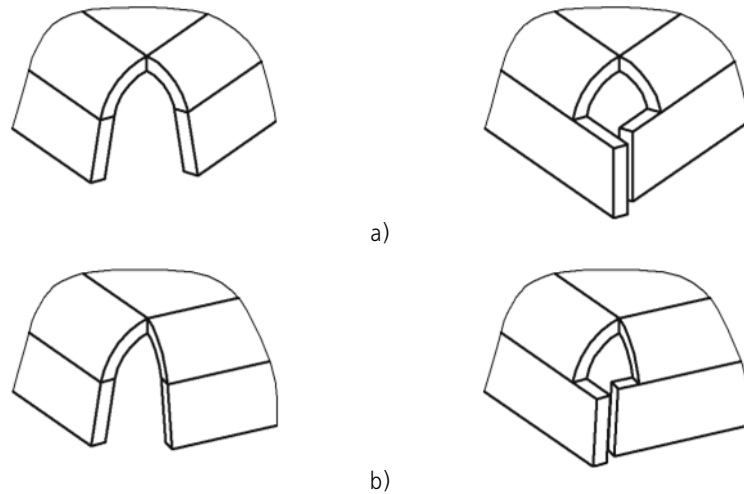


Fig. 2.4.59. Capping with overlap (original state of the part and the result of the operation):
a) closable acute angle, b) closable obtuse angle



The size of the corner being capped is affected by the relative position of the bends (the angle between the bend lines), and the angles of the bends themselves.

Capping with overlap is built as follows.

1. Straight line is constructed in the plane of measuring the gap—the trace of the outer face plane of the overlapped corner side (Fig. 2.4.60, a; 2.4.61, a).
2. The overlapping side is projected (extended) until it reaches that line (Fig. 2.4.60, b; 2.4.61, b).
3. The overlapped side is projected (extended) up until the distance between it and the overlapping side becomes equal to the specified gap.
 - 3.1. If the closable angle is acute, the gap is measured, as shown in Fig. 2.4.60, c.
 - 3.2. If the closable angle is obtuse, the gap is measured, as shown in Fig. 2.4.61, c.

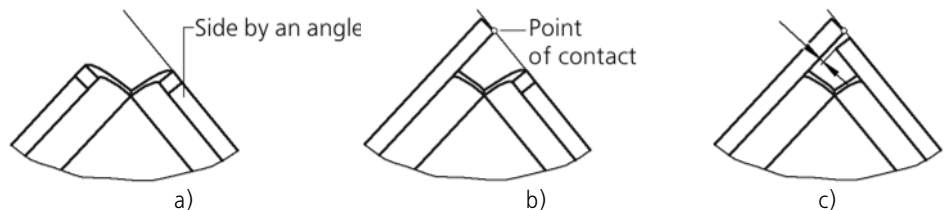


Fig. 2.4.60. Building of capping with overlap for an acute angle: a), b) definition of position for the overlapping side, c) definition of position for the overlapped side

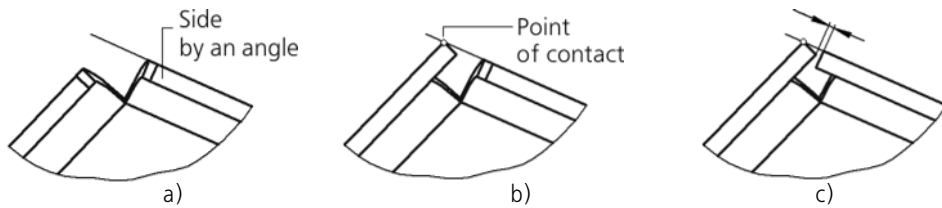


Fig. 2.4.61. Building of capping with overlap for an obtuse angle: a), b) definition of position for the overlapping side, c) definition of position for the overlapped side

The minimum gap amount is 0, the maximum is determined as shown in Fig. 2.4.62.



Fig. 2.4.62. In capping with overlap:
a) closable acute angle, b) closable obtuse angle

2.4.4.7.7. Performance of Capping



To cap one or several angles of the part, call command **Capping of Angles**.

Select a pair of adjacent bends. To do this, select the lateral face (or the edge that belongs to the lateral face) of one of them. The side faces of the selected adjacent bends and their extensions will be highlighted in the Model window (the sides of the corner being capped).

Select the capping method (see Section 2.4.4.7.1 on p. 282).

Select the angle treatment option (see Section 2.4.4.7.2 on p. 283).

Set the gap, which is used when capping the angle (see Section 2.4.4.7.3 on p. 284).

If necessary, switch on option **Continue** (see Section 2.4.4.7.4 on p. 285).

By calling the **Angle Closing** command once, you can select several angles for closing. Their list is displayed on the **Angles** bar (Fig. 2.4.63).

Closing parameters (method of closing, angle processing variant, gap and continuation) can be set up for each of the angles individually. For this purpose, select an angle in the **Angles** bar and set the required parameters.

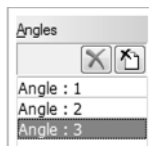


Fig. 2.4.63.



To delete the highlighted angle from the list, activate switch **Delete**.



To delete the whole list, activate switch **Exclude All**.

Having finished with corner capping settings, click the **Create Object** button on the Advanced Control Panel.

The selected angles will be closed, and the Design Tree will display the image for capping of angles.



In some cases, adjacent bends have such parameters or are positioned in such a way relative to each other, that capping the corner becomes impossible (Fig. 2.4.64).

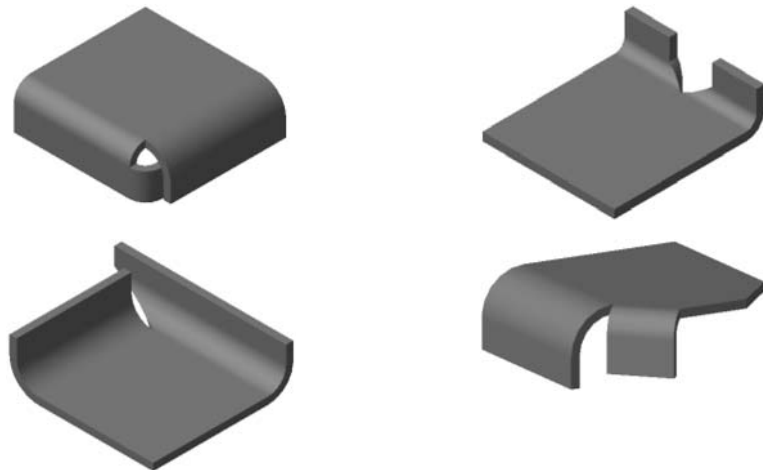


Fig. 2.4.64. Examples of adjacent bends, where the corner between them cannot be capped

2.4.4.8. Bend by sketch

Inside each sheet body you can create a bend by sketch— an element with several bends whose profile is determined by the sketch contour (Figure 2.4.65). The bend on the sketch is located along the straight edge (or several straight edges). This edge will be considered as the bend line and the edge containing it (external or internal)— base edge of the bend.

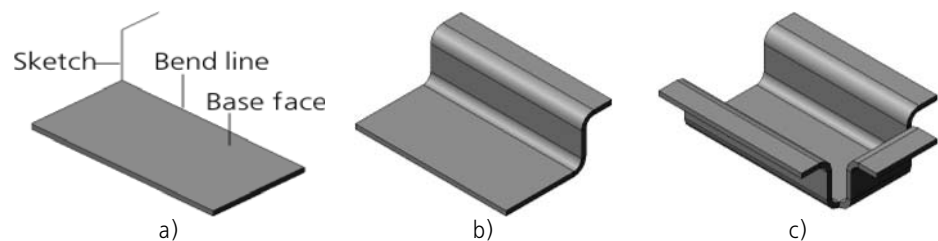


Fig. 2.4.65. ?Bend by sketch:
a) bend sketch, b) bend along one edge, c) bend along several edges



To build a bend on sketch, specify the sketch and call the **Bend on Sketch** command.

Select the bend construction method and specify an edge or a continuous sequence of edges, along which the newly created bend will be located (see Section 2.4.4.8.2).

Indicate the value and the method of defining bend radius (see Section 2.4.4.3.2 on p. 256).

Set up the definition of the flat pattern length of the bend (see Section 2.4.4.3.5 on p. 259).

Select the bend state (see Section 2.4.4.3.4 on p. 258).



Selected value and method of radius specifying as well as the broaching length definition method are used for all bends of the created element. Later on, you will be able to assign individual parameters to each of the bends (see Section 2.4.4.3.6 on p. 260).

2.4.4.8.1. Sketch Requirements

- ▼ The sketch shall be located in the plane perpendicular to the edge along which you will locate the bend.
- ▼ Sketch objects should comprise one contour.
- ▼ The Contour Must Be Open.
- ▼ The contour shall start from the point— the projection of the edge along which you will locate the bend.
- ▼ The sketch may consist of straight lines and circle's arcs only. The arcs must be connected with lines other arcs in tangent points.
- ▼ If the contour's first object is an arc, this arc shall be tangent to projection of the bend base face.



Similar to creating a sheet body by the open sketch, the contour corners form bends with user defined inner radius, and the arcs form bends with radii equal to those of the corresponding arcs.

2.4.4.8.2. Construction Method

Select creation type of bend by sketch using the **Method** group switch. The bend creation type determines bend width limits.



- ▼ If you select the **By Edge** bend creation type, the bend will be located along one edge. The bend width is equal to the edge length (Figure 2.4.66).



Fig. 2.4.66. Bend **By Edge**:
a) initial state of the model and the bend sketch, b) construction result



- ▼ If you select the **By length** creation type, the bend will be located along one edge as well. The bend width is user-defined and can be spread in any direction from the sketch plane (Figure 2.4.67). The corresponding direction of creation can be selected from the **Direction** list in the Property bar. This list offers the following options: **Direct Direction**, **Reverse Direction**, **Both Directions**, **Midplane**. The direct direction of creation is shown by the arrow on the bend ghost in the Model window. Width values to be input in the fields **Width 1** and/or **Width 2**.



Fig. 2.4.67. **By length** Bend:
a) in one direction from the sketch plane, b) in both directions



- ▼ If you select the **Chain of edges** creation type, the bend may be positioned along a chain of straight edges. All edges shall be specified as per their position order. Edges of a chain may be connected with each other through vertices (Figure 2.4.68, a) or bends (Figure 2.4.68, b).

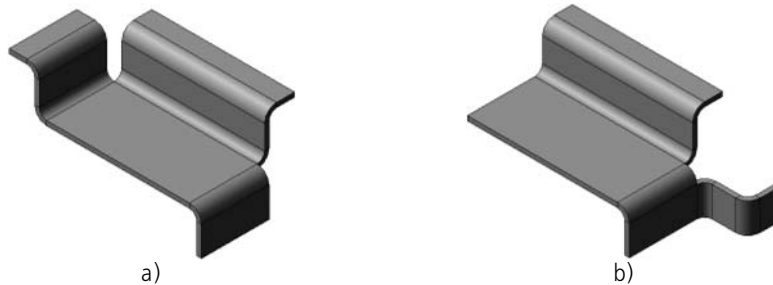


Fig. 2.4.68. Bend along a chain of edges: a) vertex connected, b) bend connected



To make a bend along the edges connected with bends, specify not only straight edges, but also arc edges mating them.

2.4.4.8.3. Cap Corners

While creating a bend by sketch, you can setup corners capping using control elements of the **Cap Corners** tab on the Property bar (Figure 2.4.69).

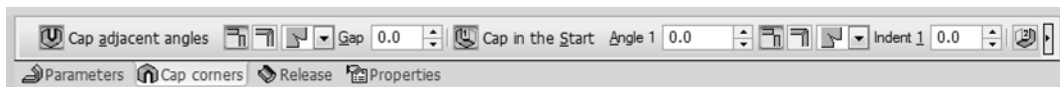


Fig. 2.4.69. **Cap Corners** Tab



The **Cap Adjacent Corners** switch is available if you create a bend along a vertex connected edge chain. This switch allows to setup corner capping at the joints of chain edges (Figure 2.4.70): to select the capping method and the corner treatment and set a distance between the corner sides. The setup procedure is similar to the **Cap** command (see Sections 2.4.4.7.1–2.4.4.7.3), with the only exception that: the **Capping with Overlap** option is not available.



Fig. 2.4.70. Bend along a chain of edges:
a) with no corner capping, b) with capping (one of the options available)



The **Cap in the Start** and **Cap in the End** switches are available if you create a bend along one edge or an open chain of edges. These switches allow to shape the first and the last edges of the bend by sketch to ensure capping with the prospective adjacent bend (for example, to join with other parts). Since there are no adjacent bends for the first and the last edge, the capping is performed with respect to the plane α , which passes through the end point *A* of the bend (Figure 2.4.71, a).

To setup the plane α position, enter the angle value between this plane and the plane perpendicular to the bend line to **Angle 1** (or **Angle 2**) field. The angle value can be both positive and negative.

If required, input to the **Offset 1** (or **Offset 2**) field a distance at which the bend edge shall be located with respect to the plane α (Figure 2.4.71, b). The offset value can be only positive; the zero offset means that the bend edge is in the plane α .

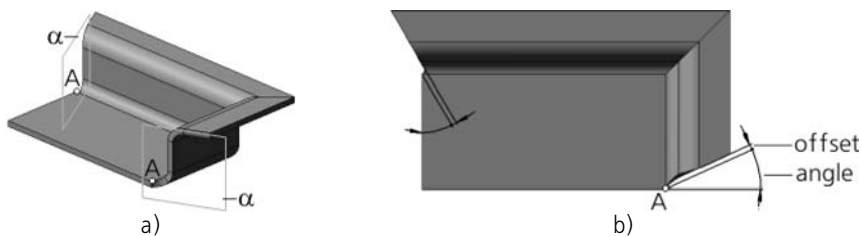


Fig. 2.4.71. Bend with closing at the beginning and at the end:
a) A point and α plane, b) top view; indent and angle

The selection of capping method and corner treatment is similar to the case of the **Cap** command (see Sections 2.4.4.7.1 and 2.4.4.7.2), with the only exception that: the **Capping with Overlap** option is not available.



Constructing a bend on sketch is impossible if negative slope is specified for closing both the beginning and the end and **Without Processing** method of processing is selected.

Please note the following specifics of constructing a bend by sketch with capping along a chain of edges. If a chain contains edges that form face internal angle (Figure 2.4.72), construction is only possible if either the **Butt Junction** or the **Chord Junction** capping options are enabled. Therewith, if the internal angle is less than 90° , the capping will be created subject to the condition that the first element on the bend sketch is an arc (Figure 2.4.73). If the first sketch element is a line, the capping is impossible.

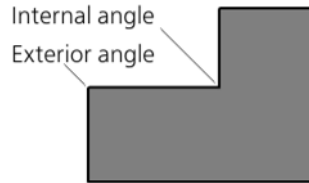


Fig. 2.4.72. Internal and external angles of the sheet body face

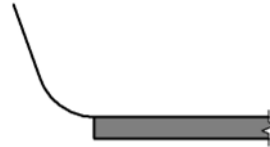


Fig. 2.4.73. Sketch starting from the arc

2.4.4.8.4. Release

When creating a bend by sketch, you can setup the bend and angle release parameters using the control elements on the **Release** tab of the Property bar (Figure 2.4.74).

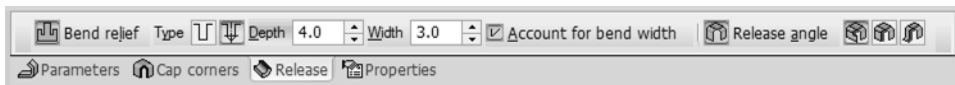


Fig. 2.4.74. **Release** Tab



The **Bend Relief** switch is available, if either **By edge** or **By length** option is enabled. Activate this switch if you need to create slots on the bend sides. Once the switch is activated, the slot shape and size control elements become available. For more information about the bend relief, see Section 2.4.4.4.5 on p. 271.



If during creation of a bend by sketch you need to delete parts of neighboring bends, activate the **Release Angle** switch and select the respective relief method. For more information about the angle release methods, see Section 2.4.4.3.3 on p. 256.

The **Bend Proper** release option can be used for deleting bend portions which connect edges of the chain (Figure 2.4.75).



Fig. 2.4.75. Bend along a chain of edges connected by bends:
a) without angle release, b) with angle release

2.4.5. Plates

Plate is a flat boss feature attached to a sheet body. A plate is created by extruding a closed sketch. The extrusion depth can be arbitrary or equal to the body material thickness.

Before adding a plate to a sheet body, you need to create a sketch in the model, that determines the shape of the plate.

2.4.5.1. Plate Sketch Requirements

- ▼ Sketch must be constructed either on the outer or on the inner flat face of the sheet body only.
- ▼ Sketch objects can be a part of one or several contours.
- ▼ Contours must be closed.
- ▼ Contours can be nested. The number of nesting levels is one.
- ▼ The outer contour must intersect with the base face contour or have common points with it.

2.4.5.2. Forming of a Plate



To build a plate, call command **Plate**.



The **Plate** command is available, if at least one sketch is selected.

The model window will display the ghost image of the created plate. The arrow shows the extrusion direction. This direction is determined by the system automatically.

The **Thickness Object** list on the **Settings** tab (Property bar) makes it possible to specify whether the plate thickness is equal to the sheet body current thickness or a user-defined constant value.

Drop down the **Thickness Object** list to select the required sheet body. Thickness of the selected body will be displayed in the **Thickness** reference field.

To specify an arbitrary plate thickness, select **Without Object** line in the **Thickness Object** list. Field **Thickness** becomes editable. Enter the desired value there.



To confirm plate creation in the sheet body, click the **Create Object** button on the Advanced Control Panel.



The model window will display the created plate, and the Design Tree the corresponding image.

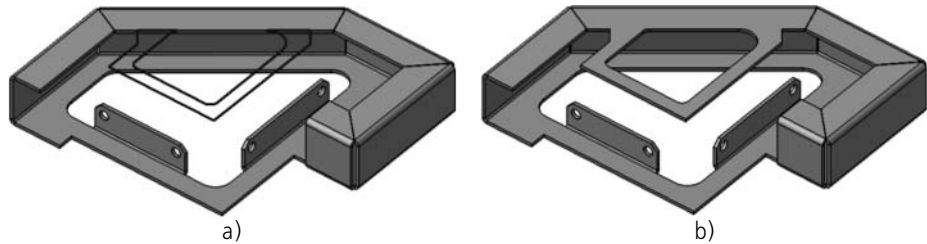


Fig. 2.4.76. Creation of a plate:
a) original state of the part and the sketch of the plate, b) operation result



If the plate fully or partially overlaps a bend, then its unbending becomes impossible. You can find more information on this in Section 2.4.7.3 on p. 307.

2.4.6. Holes



In a sheet body it is possible to create round holes and holes of a random profile— **cuts**.

To form round holes and cuts, you can use commands **Hole in Sheet Body** and **Cut in Sheet Body**, respectively. The detailed description of these commands is given in Sections 2.4.6.1 and 2.4.6.2.



To create holes in a sheet body, it is recommended to use exactly the **Hole in Sheet Body** and **Cut in Sheet Body** commands, as they take into account peculiarities typical for sheet bodies.



Cuts and holes can separate a sheet body into several parts (for more information about bodies consisting of several parts, see Section 2.3.11 on p. 224).

In each sheet body, which consists of several parts, it is impossible to create new and change existing bends, bends capping and create holes and cuts using the **Through** option. To make these operations possible, shape a body as a single piece; to do this, edit existing elements or create new elements to join all parts.

2.4.6.1. Hole



To create a round hole in a sheet body, highlight the face at which it will be located, and call command **Hole in Sheet Body**.

The model window will display the hole ghost.

By default, the hole center is positioned at the origin of the selected face's local coordinate system.

To position hole at the desired place on the face, enter the coordinates in the **p** field on the **Options** tab of the Property Bar. You can also select the hole position by the mouse. To do this, unfix the **p** field and select the desired point in the Model window. The selected point coordinates will be determined automatically.

Enter the hole diameter in field **Diameter**.





To set the hole position and diameter, handles can be used (see Section 2.1.4.2.2).





All changes are immediately reflected on the ghost of the hole in the Model window.

In group **Type** activate the switch that corresponds to the right type of building (see Table 2.4.12).

Tab. 2.4.12. Type of hole construction

Value of option	Building rules
Building type	
 By width	<p>The hole goes from the indicated face up to the opposite one in the direction, perpendicular to these faces.</p> <p>If a hole expands on a bend, possibly together with the adjacent part elements, then it wraps around this bend and the adjacent elements. You can create a hole using this method, if the following conditions are satisfied:</p> <ul style="list-style-type: none"> ▼ an outer or an inner flat face of the sheet body shall be specified for the hole construction, ▼ the hole should cross the specified face. <p>The Thickness Object list on the Parameters page of the Property Bar allows to specify whether the hole depth will coincide with the current thickness of a sheet body or will be equal to a constant value defined by the user.</p> <p>Drop down the Thickness Object list to select the required sheet body. Thickness of the selected body will be displayed in the Thickness reference field.</p> <p>If it is required to define an arbitrary thickness, select Without Object string in the Thickness Object list. Field Thickness becomes editable. Enter the desired value there.</p>
 By depth*	<p>Building of a hole by this method is cutting of the extrusion element with setting of a random extrusion distance. Extrusion is made perpendicular to the face selected for making a hole. The extrusion direction is inward with respect to the sheet body.</p> <p>To define the extrusion distance, enter the value in the Depth field on the Settings tab of the Property Bar.</p> <p>If the hole contacts the bend, it is not shifted to it, like “wrapping”.</p>

Tab. 2.4.12. Type of hole construction

Value of option	Building rules
Building type	
	<p data-bbox="372 287 501 318">Up to face*</p> <p data-bbox="601 287 1310 444">Creating a hole or cut by the 'Up to face' method amounts to cutting an extrusion feature with specifying an object for automatic extrusion distance determination. Extrusion is made perpendicular to the face selected for making a hole. The extrusion direction is inward with respect to the sheet body.</p> <p data-bbox="601 449 1310 542">To define an object to serve as an extrusion bound, select it in the Model window. This object can be represented by a face, surface, a projection or auxiliary plane.</p> <p data-bbox="601 547 1268 578">If the hole contacts the bend, it is not shifted to it, like "wrapping".</p>
<p data-bbox="355 626 1315 680">* If the file of the model, which is being edited, contains several sheet bodies, specify a sheet body in which this hole will be created. To specify the body, click it with the mouse in the model window.</p> <p data-bbox="372 686 1143 711">To change a sheet body, use the Specify Other button on the Special Control Panel.</p>	
 	<p data-bbox="372 760 1272 791">To confirm hole creation, click the Create Object button on the Advanced Control Panel.</p> <p data-bbox="372 797 1139 829">A round hole will be created in the part, and its icon in the Design Tree icon.</p>
	<p data-bbox="372 888 1315 942">Upon creating a hole in a part, a sketch is created automatically. It is located on the face, selected to construct a hole, and contains a construction point located on the hole axis.</p>

If a hole expands on the bend, then the hole will be rebuilt upon changes in the bend state (see Section 2.4.7.1 on p. 302). The resulting shape of the hole depends on its creation type and on what was the bend state at the time of creating the hole.

Let's consider examples of building and rebuilding of holes that contact bends, using the part shown in Fig. 2.4.77 as an example.

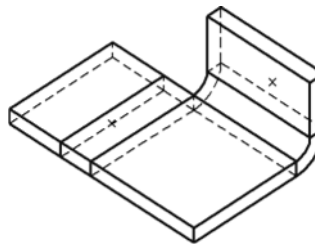


Fig. 2.4.77. Original part state. The centers of the would-be holes (positioned on invisible faces) are marked by "crosses"

Rebuilding of holes built **Through** is shown in Fig. 2.4.78.

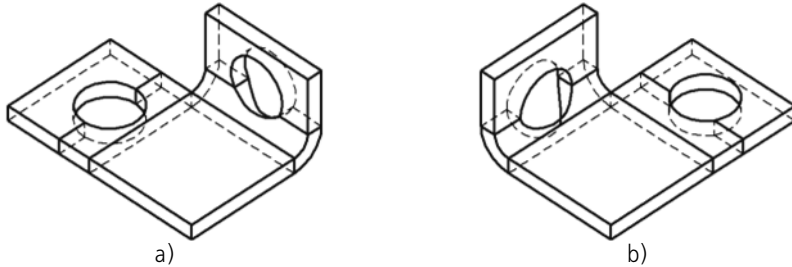


Fig. 2.4.78. Holes created by the **Through** method
a) the part view immediately after creation of holes, b) the result of changes in state of bends

Rebuilding of holes built **To depth** is shown in Fig. 2.4.79. Holes built **Up to face** are re-built in a similar way.

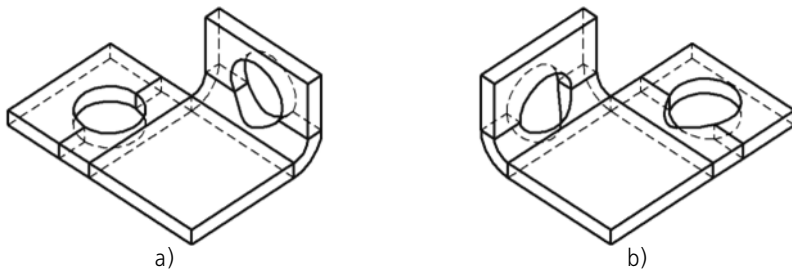


Fig. 2.4.79. Holes built by method **By depth**:
a) the part view immediately after creation of holes, b) the result of changes in state of bends

2.4.6.2. Cut

Before forming of a cut in a part, it is necessary to create at its face a sketch that depicts the profile of this cut.

2.4.6.2.1. Requirements to the Sketch

- ▼ Sketch objects can be a part of one or several contours.
- ▼ Contours must be closed.
- ▼ The sketch must be constructed either on the outer or on the inner flat face of the sheet body only (for the construction of cut by the **Through** method).

2.4.6.2.2. Forming of a Cut



Call command **Cut in sheet body**.



The **Cut** command is available, if at least one sketch (cut sketch) is selected.

The part window will display the cut ghost.

Select type of cut construction. In group **Type** in tab **Parameters** of the Properties panel activate the switch that corresponds to the right building type. Building rules for cuts are the same, as for holes (see Table 2.4.12 on p. 296).

Indicate the part section that is to be cut off during excision. To do this, activate the right switch in group **Operation Result** in tab **Excision**.



If switch **Subtraction of Element** is active, the part section located inside the surface formed by shift of the sketch of the cut, will be deleted (Fig. 2.4.80,b).



If switch **Intersection of Elements** is active, the part section located outside this surface, will be deleted (Fig. 2.4.80,c).

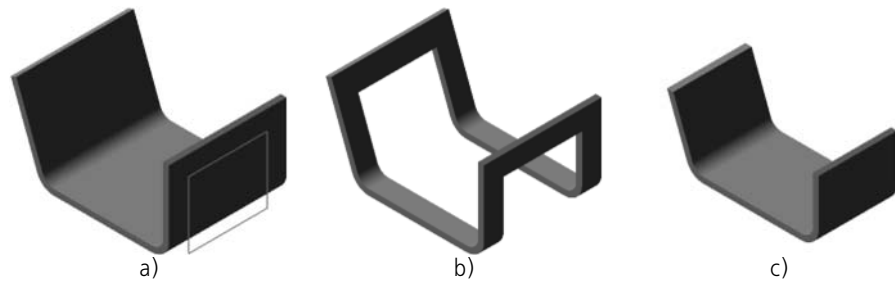


Fig. 2.4.80. The result of building of a cut in a sheet body:
a) original state of the part and the sketch of the cut, b) the cut by subtraction, c) the cut by intersection



To confirm cut creation, click the **Create Object** button on the Advanced Control Panel.

A cut will appear in the part, and its image will be displayed in the Design Tree.

If a cut expands on the bend, then the cut will be rebuilt upon changes in the bend state (see Section 2.4.7.1 on p. 302). The resulting shape of the cut depends on its creation type and on what was the bend state at the time of creating the cut.

Let's consider options of building and rebuilding of cuts that contact bends, using the part shown in Fig. 2.4.81 as an example.

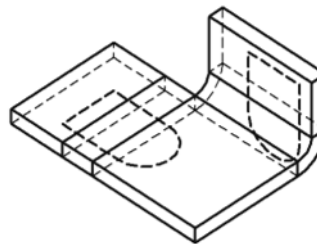


Fig. 2.4.81. Original part state. Sketches of cuts (located on invisible faces) are shown by the normal dashed line

Rebuilding of cuts built **Through** is shown in Fig. 2.4.82.

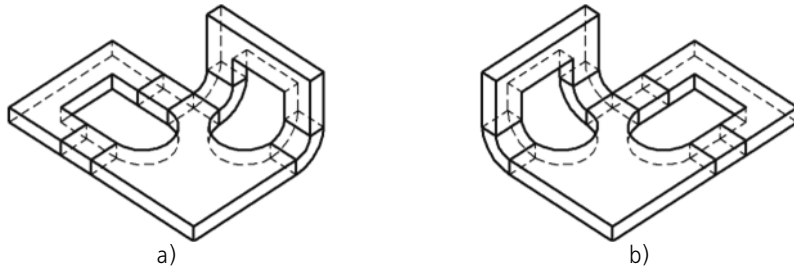


Fig. 2.4.82. Cuts created by the **Through** method:
a) the part view immediately after creation of cuts, b) the result of changes in state of bends

Rebuilding of cuts built **By depth** is shown in Fig. 2.4.83. Cuts built **Up to face**, are rebuilt in a similar way.

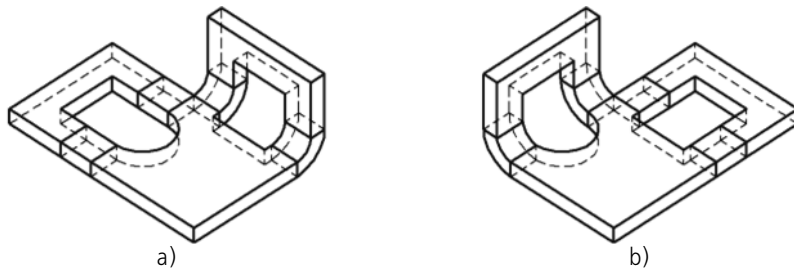


Fig. 2.4.83. Cuts built by method **By depth**:
a) the part view immediately after creation of cuts, b) the result of changes in state of bends

2.4.7. Unbending and Bending Bends. Flat pattern

Any bend (created using commands **Bend**, **Bend by Line**, **Bend by Sketch**, **Incision**, or during creation of a sheet body based on the open sketch) can be in a bent or unbent state.

By default, bends are formed in a folded state. If necessary, during creation (editing) of a bend or a sheet body it can be assigned the attribute “unbent”. This attribute can also be managed using commands **Unbend** and **Bend** in the context menu of the element (see Section 2.4.4.3.4 on p. 258).

Besides, special commands exist to change state of one or even several bends of a sheet body: **Unbend** and **Bend**.



Unbend command can be applied to bends without the “unbent” property, as well as to bends that were bent a by the **Bend** command.



Bend command can be applied to bends with the “unbent” property, as well as to bends that were unbent a by the **Unbend** command.

Working with the indicated commands is considered in detail in Section 2.4.7.1.



Assignment of the attribute “unbent” to a sheet body with several bends (for example, to an incision) means unbending of all its bends, while the **Unbend** command makes it possible to unbend bends selectively.

The described methods to manage state of bends can be applied at any time when editing the part. However, their impacts are not equal.

The difference between commands **Unbend** and **Bend** from the context menu of the sheet element and commands **Unbend** and **Bend** from menu **Operations— Features of Sheet Body** consists in the following.

The use of the first couple of commands is editing of a **separate element**: changed are the states of bends belonging to this feature, where as using the second pair means editing a whole sheet body: it gets an additional operation *Unbend* or *Bend*.

Like any other editing, sheet feature editing by the commands **Unbend** and **Bend** from the context menu could lead to **errors in objects dependent on these features** (holes, cuts, etc.).

Editing sheet elements by the **Unbend** and **Bend** commands from the menu **Operations— Features of Sheet Body** apparently cannot cause errors. Such editing, however, leads to a greater number of operations in the model and, therefore, the processing time (opening the file, rebuilding the model, creating associative views, etc.).

Therefore, if a bend does not have dependent objects (which can be checked, for example, by the command **Relations**), then its state can be changed by the context menu commands. In that case, you can better use the commands from the menu **Operations— Features of Sheet Body**.



If you know that certain bends need to be unbent to create some features, than the following work order can be preferred.

1. Create sheet features in the unbent state.
2. Create their dependent objects.
3. Bend the bends by the command **Operations— Features of Sheet Body— Bend**.

The described strategy allows avoiding an additional unbending operation.

Apart from the above-mentioned methods to change state of bends, there exists a special method to display a sheet body— **presentation in extended form**. In this mode, the bends selected by the user are shown in the bend state, while the rest are shown unbent.

States of bends set while editing the part, are ignored while switching to the extended display mode.

In the flat pattern mode you can view the part or measure its geometrical and mass-inertia properties. Editing of a part in the flat pattern mode is impossible.

Setup and activation of the mode to display the part in extended form are considered in Section 2.4.7.2 on p. 304.

2.4.7.1. Unbending and bending



To unbend or bend a bend (several bends) of a sheet body, call command **Unbend** or **Bend**, correspondingly.

After calling the command, you need to specify a **Plane of Bending/Unbending** and the **bends**, whose state needs to be changed. To do this, use the items provided on the **Options** tab of the Property Bar.

You can find more information on the purpose of elements and work with them in Sections 2.4.7.1.1 and 2.4.7.1.2.



Having finished with bend or unbend settings, click the **Create Object** button on the Advanced Control Panel.



A sheet body will be rebuilt in accordance with the preset unbending or bending parameters, while the Design Tree will display an image for unbending or bending.

2.4.7.1.1. Fixed face or plane

As a result of unbending or bending of any bend, the portion of the parts adjacent to that bend on one side is rotated with respect to the portion adjacent to the bend on the other side. In other words, one portion of the part stays still with respect to its coordinate system, while the other one moves.

When unbending/bending, you need to specify the fixed face or plane.

The fixed face is any planar face of a sheet body that belongs to its portion that stays still as a result of unbending or bending a bend (bends).

Dependence of the result of the operation on selection of the fixed face is shown in Fig. 2.4.84, using unbending of a bend as an example.

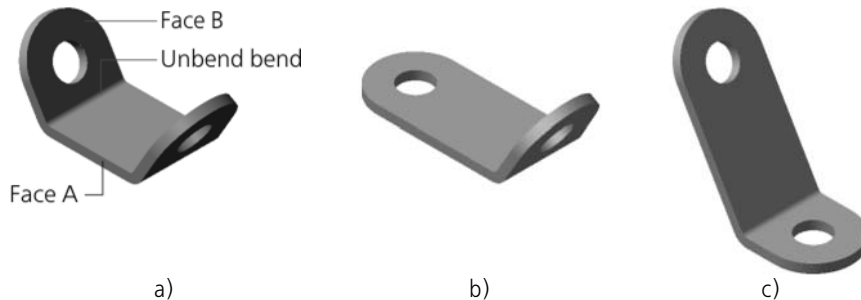


Fig. 2.4.84. Dependence of the result of unbending on selection of the bending plane:
a) original state of the part, b) face A is a plane of unbending, c) face B is a plane of unbending

Fixed plane is an auxiliary plane tangent to the bend which, when bending or unbending, will remain fixed and where the unbent bend (bends) will be located.

Objects that can be selected for setting the fixed plane:

- ▼ The bend's edge (cylindrical, conical)— external edge of the bend is specified. Auxiliary plane is created automatically and located at one of vertices of the face (Fig.2.4.85)
- ▼ The auxiliary plane tangent to the bend's edge (cylindrical, conical) of the sheet body.



The auxiliary plane with the required position may be created by the user in advance using commands **Tangent plane** or **Plane tangent to the edge in the point** on the **Auxiliary geometry** bar.

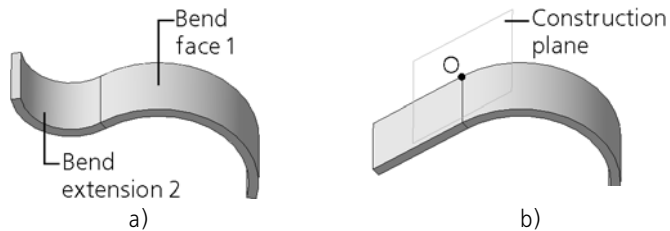


Fig. 2.4.85. Unbending of a sheet body:
a) initial state of a part, b) the result of unbending of the bend (when selecting a bend 1)



To specify the fixed face or plane, enable the **Plane** switch and select the required object in the model window.

When unbending a body, once the bending face has been specified, the automatically created auxiliary lane appears in the model.



If an auxiliary plane had been used for unbending, to put the body in the initial (bent) state, specify this plane as a fixed one using command **Bend**.

The selected plane will be highlighted in the Model window and in the Design Tree. The selected face will be highlighted in the Model Window, while the element it belongs to will be highlighted in the Design Tree.

2.4.7.1.2. Selection of Bends



To proceed with selecting bends, whose state needs to be changed, activate the switch **Select Bends**.

- ▼ To use the unbending command, you need to select the bent bends. To select a bent bend, pick any of its cylindrical faces in the Model window.
- ▼ To use the bending command, you need to select the unbent bends. To select an unbent bend, pick any its flat face that is supposed to become cylindrical, in the Model window.



If a model contains several sheet bodies, only the bends which belong to the same body as the auxiliary plane or the face indicated as the fixed one will be available.

The model window will display the selected bends as highlighted, and the Design Tree will show bends and sheet elements, to which they belong.

The list of bends indicated for unbending (bending), is displayed in panel **Bends** (Fig. 2.4.86). This panel also contains buttons to manage the list of bends. The functions of the buttons are represented in Table 2.4.13.

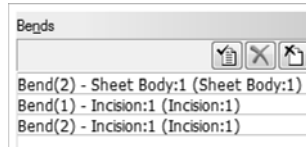


Fig. 2.4.86.

Tab. 2.4.13. Management of the list of bends

The	Description
	<p>Select all</p> <p>Allows selecting all bends.</p> <p>When working with the unbending command, all ‘bent’ bends are included in the list, while when working with the bending command—all ‘unbent’ ones are included.</p> <p>After the button is pressed, the selected bends are highlighted in the model window and in the Design Tree.</p>
	<p>Delete</p> <p>Allows excluding a selected bend from the list.</p> <p>Upon clicking the button, the selected bend becomes dehighlighted in the Model Window.</p> <p>Selecting a selected bend in the Model Window results in excluding it from the list.</p>
	<p>Exclude All</p> <p>Clears the list of bends.</p> <p>Upon clicking the button, all bends become dehighlighted in the Model Window.</p>

2.4.7.2. Flat pattern

Before switching to the mode of displaying the flattened sheet body, you need to set up the **flat pattern parameters**— select the fixed face or plane and define bend states (see Section 2.4.7.2.1).

Having finished setup of flat pattern parameters, save parameters by pressing button **Create Object** in the Advanced Control Panel.

To switch to the extended display mode, call command **Flat Pattern**. The current model will be displayed according to its stored flat pattern parameters.

For the convenience of viewing the flat pattern of the part, you can use the **Flat Pattern** orientation (see Section 2.4.7.2.2 on p. 306).

To switch to the normal mode of working with the sheet body, call the command **Flat Pattern** again.

2.4.7.2.1. Flat Pattern Parameters

To set flat pattern parameters, call command **Flat Pattern Parameters**. The Property Bar will display controls for setting up a flat pattern (Fig. 2.4.87).

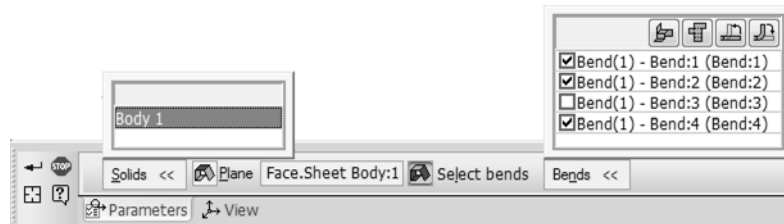


Fig. 2.4.87. Flat pattern parameters control elements

- ▼ If a model contains several sheet bodies, flat pattern parameters shall be setup for each body. To setup the body flat pattern parameters, select this body in the body list and specify its fixed face or plane and unbent bends.
- ▼ If a model contains only one sheet body, there is no need to select it; select directly the fixed face or plane and bends.

Selection of Bodies

To select a body, specify it in the list on the **Bodies** bar (Figure2.4.87). The selected body will be highlighted in the Model window.

The list lines are displayed in different colors:

- ▼ the line of a body for which unfold settings have not been defined yet has read color,
- ▼ the line of a body for which unfold settings have already been defined has green color.



There is no need to define settings for all bodies, you can setup one or several bodies.

Selection of the fixed face or plane

Selection of the fixed face or plane is performed in the same way, as when working with bending and unbending commands (see Section2.4.7.1.1 on p. 302).



Faces that do not belong to the sheet body or the planes not related to the body, whose flat pattern parameters are being setup at the moment, are unavailable for selection.

Bends Selection



To define the states of bends, in which the latter will stay through the flat pattern mode, activate the switch **Select Bends**.

After that the panel will become available (Fig.2.4.87), that contains the list of all the bends existing in the current part.





Options are displayed to the left from the bend names. Bends with enabled options will be displayed as unbent in the flat pattern mode, and bends with disabled options — bended. To cancel the bend, enable the option near its name. When selecting a bend in the list, it is highlighted in the Model Window, and upon selecting it in the Model Window— is highlighted in the list. This helps control correctness of settings.



Bends that do not belong to the sheet body, whose flat pattern parameters are being setup at the moment, can not be specified.

To speed up the setup, use the buttons located above the list of bends. The description of buttons is given in Table 2.4.14.

Tab. 2.4.14. Managing bend states

	The	Description
	Bend all	Serves to disable options of all bends.
	Unbend all	Serves to enable options of all bends from the list.
	Bend selected	It allows to disable options of the selected bends*.
	Unbend selected	Serves to enable options of the bends selected in the list*.

* To select multiple bends in the list, use the keys *<Shift>* and *<Ctrl>*.
To select multiple bends in the Model Window, simply select them one after another.

Face or plane Selection for Flat Pattern Orientation

By default, the **Flat Pattern** orientation (see Section 2.4.7.2.2) is set in such a way that the gaze direction is perpendicular to the selected fixed face or plane (if there are several bodies, to the fixed face or plane of the body, whose flat pattern parameters were setup first).

If required, you may select any other face or plane for the **Flat Pattern** orientation. To do this, activate the **View** tab of the Property bar and then select the required face or plane in the Model window. The face will be highlighted, while its name will appear in the **View's Plane** field.

2.4.7.2.2. Flat Pattern Orientation

When flat pattern parameters are set, the **Flat pattern** orientation is added for the current model's list of orientations (see Section 2.1.3.2 on p. 121).

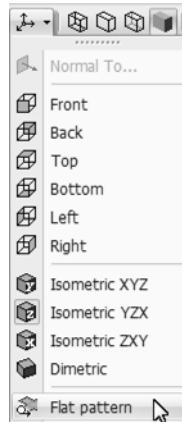


Fig. 2.4.88. **Flat Pattern** orientation in the list of orientations

In the **Flat Pattern** orientation, the model shall be positioned in such a way that the gaze direction is perpendicular to the face or auxiliary plane specified as the view's plane when setting the flat pattern parameters (see Section 2.4.7.2.1).

2.4.7.2.3. Deleting Flat Pattern Parameters

To delete flat pattern parameters, call the command **Operations— Features of Sheet Body— Delete Flat Pattern Parameters**. The earlier defined parameters of the flat pattern and **Flat Pattern** orientation will be removed from the current model.

If the sheet body flat pattern parameters in the current model have not yet been set up, then those cannot be deleted. In this case, command **Operations— Features of sheet body— Delete Flat Pattern Parameters** is not available.

2.4.7.3. Peculiarities of Unbending and Bending



When unbending and bending bends using the **Unbend** and **Bend** commands, as well as when using the **Flat pattern** command, the following should be taken into account.

- ▼ If the bend is not involved with any other features, then its unbending and bending is always possible.
- ▼ Elements that contact the bend, can be located in such a way, that unbending or bending of this bend will prove to be impossible (Fig. 2.4.89, 2.4.90).

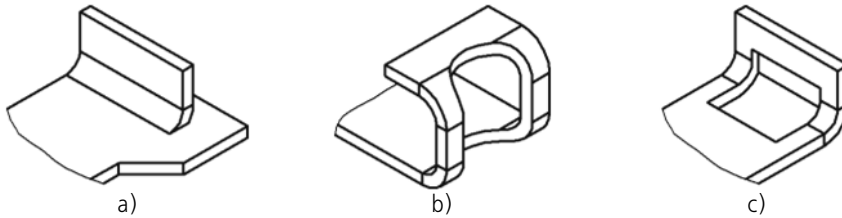


Fig. 2.4.89. Examples of sheet elements that make impossible unbending of the bend:
a) plate, b) hole, c) cut

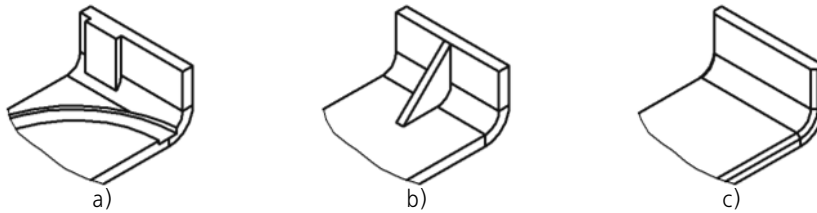


Fig. 2.4.90. Examples of shape-generating and construction elements of a part,
that make impossible unbending of the bend:
a) glued and cut out elements, b) reinforcement plate, c) facet

- ▼ The operations, that result in bend edges or faces being totally rebuilt, always make its unbending or bending impossible (Fig. 2.4.91).



Fig. 2.4.91. Examples of operations that make impossible unbending of the bend:
a) casing, b) slope

- ▼ If a hole or cam contour cut expands on the bend cylindrical part and adjacent part of a sheet body, passing non-perpendicular to this flat face, unbending is not available.



Fig. 2.4.92. Example of holes and cuts that make impossible unbending of the bend

- a) Holes built by method **By depth**:
 b) Cuts built by method **Up to Face** (cut sketch on the upper flat face of the body is shown by the dashed line)

- ▼ If a sheet body comprises several parts (for example, if a body is divided by the cut operation), change of the bends state is not possible. To unbend or bend the bends, modify a body to make it a single piece by editing existing elements or creating new elements to join all parts.

2.4.7.4. Flat Pattern Drawing

Associative views of KOMPAS-3D drawings support creation of flat pattern images of sheet bodies according to flat pattern parameters, stored within those models.

A flat pattern image can be produced when creating the following associative views:

- ▼ Arbitrary View,
- ▼ Projection View,
- ▼ View in Arrow Direction.



This section is intended for users with experience in creation of associative types. Building of these types is not considered in detail in the given section. If you don't master methods of work with associative types, it is recommended to see Section 3.6 on p. 1205.



To generate a flat pattern image in the view being created, activate the **Flat Pattern** switch on the **Options** tab of the Property bar. It is accessible if the flat pattern parameters are set up in the model.

Management of drawing of lines of a bend is performed in tab **Lines** (see Section 3.6.1.3.2 on p. 1211). Bend lines can be drawn automatically **if the plane of this view projections is parallel to the flat faces produced when unbending bends.**

To correctly select the plane of the view projections containing the flat pattern, you need to know how the flattened part will be positioned with respect to the main projection planes. This position depends on what face of the body or plane was picked as the fixed one at the time of setting up the flat pattern.

Let's consider selection of a projection plane to display the flat pattern for a part shown in Fig. 2.4.93, a.



Fig. 2.4.93. Sheet Body: a) in projection **Isometric projection XYZ**, b) master view

Let's assume that the body master view (Fig. 2.4.93, b) has already been built in the drawing. It is necessary to create a flat pattern.

In such a case, it is helpful to create an arbitrary view of this model in the drawing.

When creating an arbitrary view, the line **Flat Pattern** becomes available in the **Orientation** list on the **Options** tab of the Property Bar (Fig. 2.4.94). Selecting it means having the plane of the view projections parallel to the face selected as the fixed one at the time of setting up the flat pattern of the model.

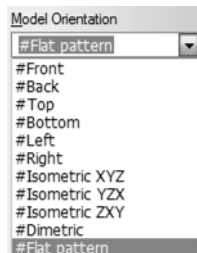


Fig. 2.4.94. **Flat pattern** in the list of model orientation on arbitrary view

If face A was set as the fixed face (see Fig. 2.4.93, a), to create an image of the flat pattern it is necessary to select orientation **Flat Pattern**.

If the fixed face was defined by Face B, then, to create the flat image, select the orientation **Top**, **Bottom** or **Flat Pattern**.

If the fixed face was set to Face C, then, to create the flat image, select the orientation **Top** or **Bottom**.



Switch on formatting, set up other view parameters, and then confirm its creation.



An arbitrary view containing the flat pattern image is automatically assigned a notation containing an “unbent” graphical symbol and a scale if the latter is different from the scale indicated in the title block.

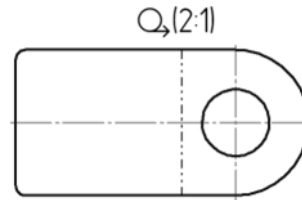


Fig. 2.4.95. Flat pattern of a sheet body



You do not have to use the automatically generated **Flat pattern** orientation. If necessary, you can manually create the desired orientation in the model and use it when creating a flat pattern in the drawing.

When creating a View by Arrow or a Projection View, then you cannot select an orientation from the list, since the position of the projection planes of those views depends on the viewing direction. When selecting that direction, you need to take into account the position of the flattened part.



A sheet body is displayed on cut or section views in the same state as on the base view. When a part state is changed on that view (flattening being turned on or off), the cuts (sections) are rebuilt automatically.

2.4.8. Stamped Features

When creating a sheet body in KOMPAS-3D, it is possible to build the following stamp elements:

- ▼ Open and closed stamping (Fig. 2.4.96, a and 2.4.96, b)
- ▼ Beat (Fig. 2.4.96, c)
- ▼ Louver (Fig. 2.4.96, d).

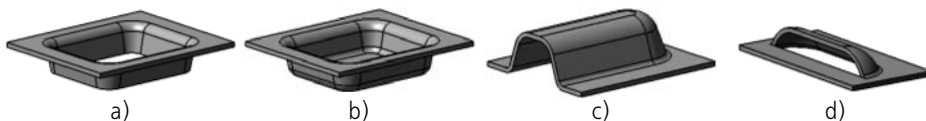


Fig. 2.4.96. Stamped features

In fact, a drawn cutout creation is not a bending operation; rather, it is a deforming operation where the sheet material is stretched and its thickness reduced. When building stamp elements in a sheet body KOMPAS-3D, this change of material thickness is not counted.

It is impossible to unfold bends of stamp elements.

General methods to build stamp elements are described in Section 2.4.8.1, and special methods in Sections 2.4.8.2– 2.4.8.4.



To set parameters for stamp elements, handles can be used (see Section 2.1.4.2.2).

2.4.8.1. General Building Methods

After calling the command to build a stamp element, the Properties panel displays control elements that allow to set up various parameters of the element. The following parameters are common for all the stamp elements:

- ▼ Direction,
- ▼ Base rounding,
- ▼ Saving of settings.

2.4.8.1.1. Direction of building

The face of a sheet body that contains the sketch of a stamp element, is considered as the **basic face** of this element. No matter at which face— external or internal— the sketch is located, the stamp element based on it can be directed both inside of the part and outside of it (Fig. 2.4.97). In this case, thickness of sheet material is not counted, owing to which geometrical parameters of the element don't depend on the direction of building.

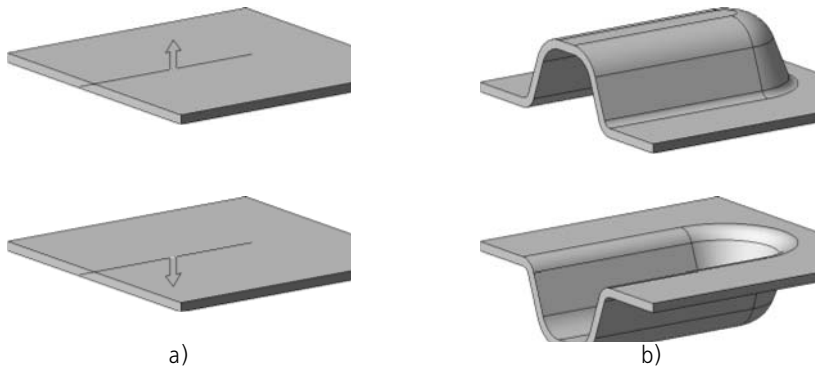


Fig. 2.4.97. The direction of building of a stamp element:
a) selection of direction, b) the result of building



The forward construction direction is the outward direction from the base face, the reversed one is inward. The current direction of creation is indicated on the screen by a ghost arrow.

To indicate the side of the basic face, at which the beat will be located, activate the right switch in group **Direction of Building**.

2.4.8.1.2. Rounding of Base Edges

Base of a stamped feature is a portion of a sheet body where the stamp feature is connected with its adjoining flat elements.

base edges are where side wall faces of a stamped feature meet the faces of its adjoining flat elements of the sheet body. In this case, the edges that belong to inner side faces of a dimple

or drawn cutout feature are considered **inner base edges** of the bend, while the ones that belong to the outer side faces are the **outer base edges** (Fig.

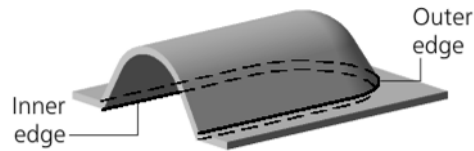


Fig. 2.4.98. Inner and outer base edges

Dimple or drawn cutout features can be created with base edge rounding or without rounding. To round base edges, enable the **Base Rounding** option and enter the rounding radius in the field **Radius of Base Round**. The specified radius value R is used to round outer edges of the base. The rounding radius of inner edges is equal to the sum $(R + S)$, where S is the thickness of the sheet material. The minimum value of base edge rounding radius is 0.

2.4.8.1.3. Saving of Settings

You can save the preset parameters for the stamp element for their further use when building similar elements up to the end of the work session. To do this, after completing element parameter setup, switch on option **By default**.

2.4.8.2. Stamping



The present section describes the procedure to build open and closed stamping. They have practically the same set of parameters and are created in a very similar way.

To build open stamping, call command **Open Stamping**.

To build closed stamping, call command **Closed Stamping**.

The commands to build stamping are available, if one sketch is highlighted— stamping profile (see Section 2.4.8.2.1).

Indicate the building direction (see Section 2.4.8.1.1 on p. 312).

Select the fixed side of stamping (see Section 2.4.8.2.2).

Set stamping height (see Section 2.4.8.2.3).

Set parameters for lateral walls (see Section 2.4.8.2.4).

Set parameters for edge rounding (see Sections 2.4.8.1.2 on p. 312, 2.4.8.2.5 and 2.4.8.2.6).



If a dimple is created with rounding bottom and/or base edges, then it is impossible to simultaneously satisfy the specified angle, radii and height values in the cases of small (relative to radii) heights of that dimple or drawn cutout. Since the height and the radius have a higher priority, the value entered in the **Angle** field will be ignored. The slope angle of side walls is calculated by the system automatically (this is, in fact, the inclination angle of lines tangent to side walls, since their planar portions become degenerate in these cases).



Having finished with settings, click the **Create Object** button on the Advanced Control bar to confirm the changes.

A new feature will be added to the sheet body being developed, and the respective icon will appear in the Design Tree:



▼ Open stamping,



▼ Closed stamping.

2.4.8.2.1. Stamping Profile

The **profile of a dimple or drawn cutout** is the shape of its bottom. The profile is defined by a sketch on which a dimple or drawn cutout is based. A thin-wall feature produced by extruding a sketch in the creation direction makes the side walls of a dimple or drawn cutout.

Requirements for the Sketch

- ▼ Sketch must be constructed either on the outer or on the inner flat face of the sheet body only.
- ▼ Sketch objects should comprise one contour.
- ▼ The contour can be closed or open.
- ▼ If the contour is closed, then it can intersect with the base face or fully belong to it.
- ▼ If the contour is open, then it must intersect the base face so as to have two common points with the edges that make up its outer contour. The end points of the contour may belong to those edges or be outside the base face.
- ▼ The common points of the contour and the base face shall not coincide with the vertices of the edges surrounding the base face.
- ▼ The contour shall not intersect or have common points with other body elements.


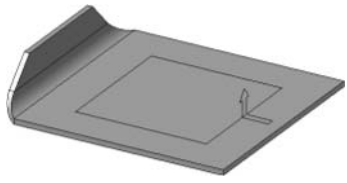
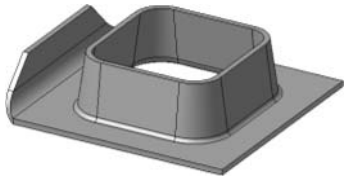

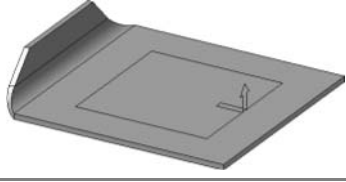
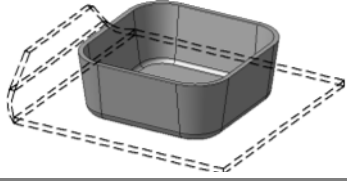
2.4.8.2.2. Fixed Side

Fixed side is the portion of the base face, whose position does not change when constructing a dimple or drawn cutout. The fixed side of a face may be inside the profile of a dimple or drawn cutout or outside it. The ghost arrow indicating the construction direction starts precisely at this location on the face.


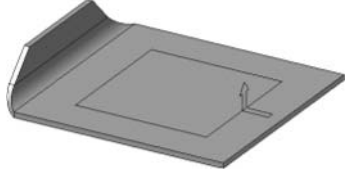
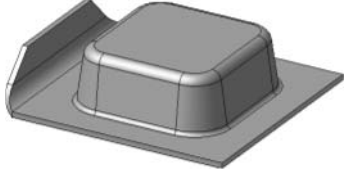

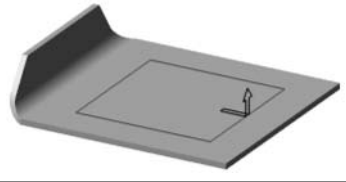
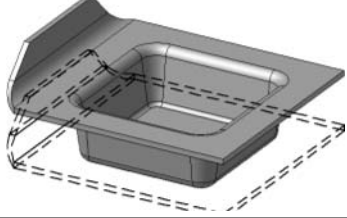
To change the fixed side, use the **Side** switch group. When changing the fixed side, the ghost arrow changes accordingly in the Model window.

Dependence of the result of building of open stamping on selection of the fixed side is given in Table 2.4.15, and closed stamping— in Table 2.4.16. A stroked line shows the original state of a part in figures. Note that closed stamping always has both a bottom and a base, while open stamping has either a bottom, or a base (depending on selection of the fixed side).

Tab. 2.4.15. Fixed side of a dimple

	Value of option Side	Original part state	The result of creation of stamping
	Side 1		
	Side 2		

Tab. 2.4.16. Fixed side of a dimple

	Value of option Side	Original part state	The result of creation of stamping
	Side 1		
	Side 2		

2.4.8.2.3. Height

To define stamping height, it is necessary to select the method to set stamping height and enter the height value.

When building open stamping, two methods to set height are available:



▼ **Full,**



▼ **Outside.**

When building closed stamping, three methods to set height are available:



▼ **Full,**



▼ **Inside,**



▼ **Outside.**

Select the method of defining the height, and then enter the desired value in the **Height** field.



When using the **Full** method, the specified value of the dimple height must be greater than the thickness of the sheet body.

2.4.8.2.4. Lateral Walls

When building stamping, you can manage the following parameters for lateral walls: direction to add material and slope.

Direction to add material

The width of a dimple or drawn cutout side walls can be counted inward or outward with respect to the surface swept by the profile moved in the creation direction.

To change the direction of adding side wall material, use the **Side Walls** switch group:

▼ For open stamping:



▼ **Inside**



▼ **Outside**

▼ For closed stamping:



▼ **Inside**



▼ **Outside**

Slope of Lateral Walls


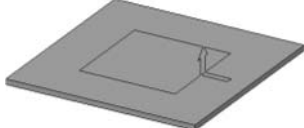
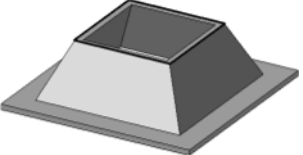


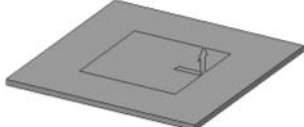
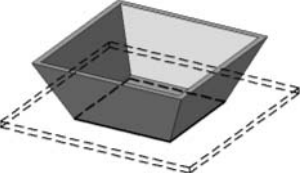
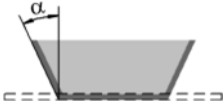
Lateral walls of stamping can be sloped in the direction of building. To build stamping with slope of lateral walls, enter the value for slope angle in field **Angle**. Zero value in this field stands for absence of slope.

The side walls inclination of a dimple or drawn cutout is made according to the following rules.


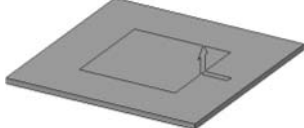
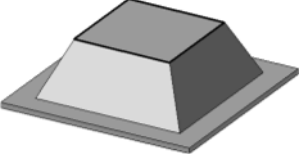


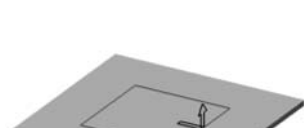
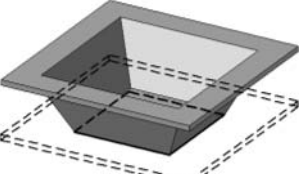
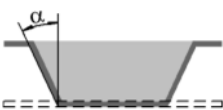
- ▼ The inclination angle is counted from the normal to the base face.
- ▼ The angle counting direction is selected such as to have side walls inclined inward with respect to the bottom of a closed stamping or open stamping.
- ▼ Original profile dimensions (defined by the sketch) are maintained near the bottom of a closed stamping or open stamping.

Examples to build open and closed stamping with slope of lateral walls are given in Tables 2.4.17 and 2.4.18, respectively. Direction to add material of lateral walls in all the examples—inside; α — draft of side walls; a stroked line indicates the original position of a part, while a thickened one shows the face that has preserved a shape and dimensions of the profile as a result of building.

Tab. 2.4.17. Open stamping side wall taper

	Original part state; Position of the fixed side	Construction result	Section by a plane perpendicular to the basic face
			
			

Tab. 2.4.18. Closed stamping side wall taper

	Original part state; Position of the fixed side	Construction result	Section by a plane perpendicular to the basic face
			
			

2.4.8.2.5. Rounding of Lateral Edges

Side edges are the edges at the joints of side wall faces of a dimple or drawn cutout. In this case, the edges that belong to inner side faces of a dimple or drawn cutout are considered **inner side edges** of the bend, while the ones that belong to the outer side faces are the **outer side edges** (Fig.2.4.99).

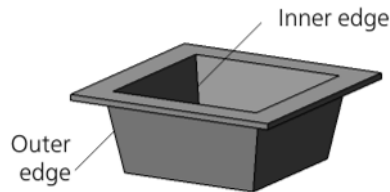


Fig. 2.4.99. Inner and outer side edges a dimple or drawn cutout

A dimple or drawn cutout can be created with or without side edge filleting.

To round side edges, enable the **Edge Round** option and enter the rounding radius in the **Radius** field. The specified radius value R is used to round inner side edges. The rounding radius of inner edges is equal to the sum $(R + S)$, where S is the thickness of the sheet material. The minimum radius value of side edge filleting is 0.

If lateral walls of stamping are sloped (see Section 2.4.8.2.4), R and $(R + S)$ are minimal radii of rounding for lateral edges. These radius values are maintained throughout the bottom of a dimple or drawn cutout. For example, the Fig.2.4.100 shows a drawn cutout with a slope on side walls and a round on side edges.



Fig. 2.4.100. A drawn cutout with a slope on side walls and a round on side edges: a) the rounding radius is equal zero, b) the rounding radius is greater than zero



The side edges of a dimple or drawn cutout, which are made by smoothly adjoining faces, are not rounded.

2.4.8.2.6. Rounding of Edges at the Bottom

Bottom edges are the ones where the dimple bottom faces meet its side wall faces. In this case, the edges that belong to inner side faces of a dimple or drawn cutout are considered **inner bend edges** of the bottom, while the ones that belong to the outer side faces are the **outer bend edges** of the bottom (Fig. 2.4.101).

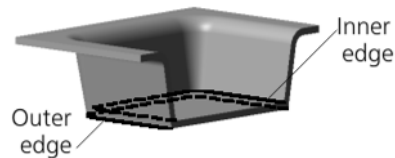


Fig. 2.4.101. Inner and outer bottom edges of a dimple or drawn cutout (the sheet metal model is cut by a plane perpendicular to the base face)

A dimple or drawn cutout can be created with or without bottom edges filleting.

To round bottom edges, enable the **Bottom Rounding** option and enter the rounding radius in the **Bottom Rounding Radius** field. The specified radius value R is used to round inner edges of the bottom. The rounding radius of inner edges is equal to the sum $(R + S)$, where S is the thickness of the sheet material. The minimum value of bottom edge rounding radius is 0.



Open stamping can have either a bottom, or a base, depending on selection of the fixed side (see Table 2.4.15 on p. 315). Therefore, during its creation the Properties panel doesn't include control elements for rounding of the bottom or the base (see Section 2.4.8.1.2 on p. 312).

2.4.8.3. Beat

In a sheet body KOMPAS-3D a beat can be created with one of the three section shapes:

- ▼ Round (Fig. 2.4.102, a),
- ▼ U-shaped (Fig. 2.4.102, b),
- ▼ V-shaped (Fig. 2.4.102, c).

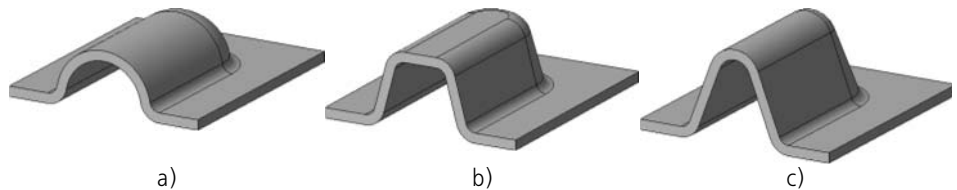


Fig. 2.4.102. Beat section shapes

Fig. 2.4.103.



To build a beat, call command **Beat**.

The command is available, if one sketch is highlighted—the beat sketch. The sketch defines the beat configuration and position. Requirements to the sketch are enumerated in Section 2.4.8.3.1.



A beat cannot be created if its sketch has common points with edges adjoining to bends.

Indicate the building direction (see Section 2.4.8.1.1 on p. 312).



Select beat section shape: **round**, **U-shaped** or **V-shape**.

Select beat ends processing type (see Section 2.4.8.3.2).

For convenience of work, panel **View** indicates a beat pattern (see Section 2.4.8.3.3).

Select the method to build a beat (see Section 2.4.8.3.4).

Enter beat parameters in the respective fields on the Property Bar. The set of these fields depends on the selected creation method.

Set up rounding parameters for edges at the base (see Section 2.4.8.1.2 on p. 312). For a U-shaped beat it is also possible to round edges at the bottom. This is done in the same way, as during creation of stamping see Section 2.4.8.2.6 on p. 318.



If, when creating a V-shaped or U-shaped beat with bottom and/or base edges rounding, a too small height is defined (compared to the radii), then it is impossible to simultaneously meet the values of the angle, radius (radii) and height. Since the height and the radius have a higher priority, the value entered in the **Angle** field will be ignored. The slope angle of side walls is calculated by the system automatically (this is, in fact, the inclination angle of lines tangent to side walls, since their planar portions become degenerate in these cases).

If the beat doesn't fit in completely within the basic face, it is cut by the plane (planes) of the corresponding end face (Fig. 2.4.104).



Fig. 2.4.104. The beat exceeding the limits of the basic face:
a) the beat sketch; b) the result of building



Having finished with settings, click the **Create Object** button on the Advanced Control bar to confirm the changes.



A new feature will be added to the sheet body being developed, and the respective icon will appear in the Design Tree:

2.4.8.3.1. Requirements to the Sketch

- ▼ Sketch must be constructed either on the outer or on the inner flat face of the sheet body only.
- ▼ Sketch objects can be a part of one or several contours.
- ▼ The contours can be closed or open.
- ▼ If a contour consists from several graphical objects, then those must smoothly adjoin to each other.
- ▼ Contours may intersect each other, but a contour self-intersection is not allowed.

2.4.8.3.2. Treatment of Ends

End of beat is a beat element corresponding to an end point with a contour in the sketch. The following options are available for treatment of ends:

- ▼ Closed (Fig. 2.4.105, a),
- ▼ Open (Fig. 2.4.105, b),
- ▼ Chopped (Fig. 2.4.105, c).

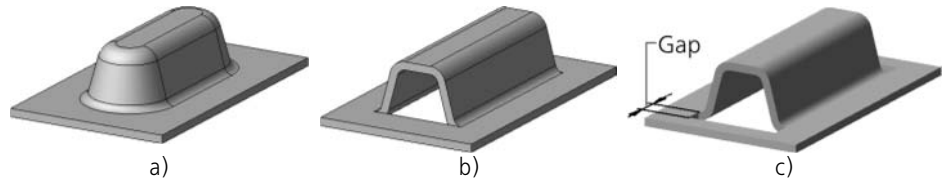


Fig. 2.4.105. Beat ends processing types

The selected type is applied to all ends of the current beat.

When selecting the **Chopped** processing type, a **Gap** field appears on the Property Bar, in which you need to enter the gap amount of chopping.

2.4.8.3.3. View Sample

The beat image is shown in panel **View**. There are two displaying options:


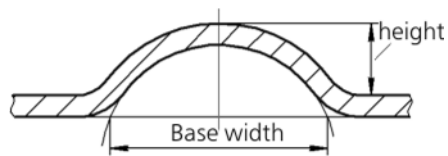
- ▼ A **side crosscut** with the dimensions corresponding to the section parameters,
- ▼ A **sample** — a schematic image of a rectangular beat that accounts for the selected section shape, ends processing type and base rounding.

To select the image type in panel **View**, option **Pattern** is used. If it is enabled, then the sample beat is displayed on the preview pane, if disabled— then it is the cut.

2.4.8.3.4. Construction Method

To select the method to build a beat, list **Method** is used. The set of methods available in this list, depends on the selected section shape (see Table 2.4.19–2.4.21).

Tab. 2.4.19. Method of creating a round beat

Method	Parameters
 By height and base width	


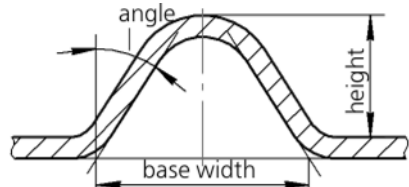

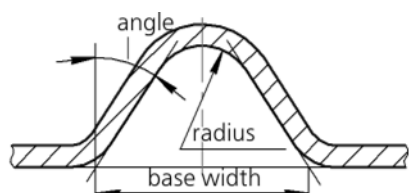

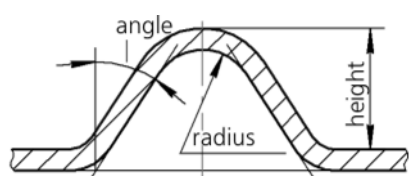

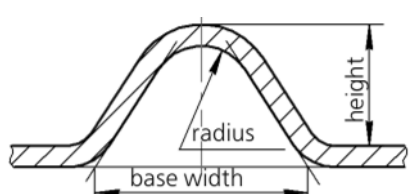
Tab. 2.4.19. Method of creating a round beat

Method	Parameters
A	By height and radius
D	By radius and base width

Tab. 2.4.20. Methods to build a U-shaped beat

Method	Parameters
D ₁	By height, angle and base width
D ₂	By width of the base, by height and width of the bottom
D ₃	By width of the base, by angle and width of the bottom
D ₄	By angle, height and bottom width

Tab. 2.4.21. Methods to build a V-shaped beat

	Method	Parameters
	By height, angle and base width	
	By height, angle and radius	
	By radius, angle and base width	
	By height, radius and base width	

2.4.8.4. Louver

Serves to create a louver by a straight line in a sheet body. Two louver types are available:

- ▼ Elongated (Fig. 2.4.106, a),
- ▼ Cut (Fig. 2.4.106, b).

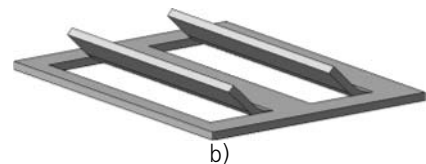
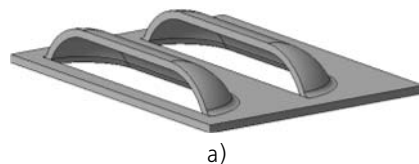


Fig. 2.4.106. Louver types



To build a louver, call command **Louver**.

The command is available, if one sketch is highlighted— the louver sketch. Requirements to the sketch are enumerated in Section 2.4.8.4.1.



Indicate the building direction (see Section 2.4.8.1.1 on p. 312).



Select Louver position— to the right or to the left relative to the section in the sketch, activating the right switch in group **Position**.

Set the louver height (see Section 2.4.8.4.2).

Select the method to build a louver (see Section 2.4.8.4.4).

Set the louver width (see Section 2.4.8.4.4).

Set up rounding parameters for edges at the base (see Section 2.4.8.1.2 on p. 312).



Having finished setup, press button **Create Object** in the Advanced Control Panel to confirm creation of the louver.



A new element (elements) with the preset parameters will be added to the designed sheet body, while the Design Tree will display the corresponding image.

2.4.8.4.1. Requirements to the Sketch

- ▼ Sketch must be constructed either on the outer or on the inner flat face of the sheet body only.
- ▼ A sketch may contain a single line segment or several lines.
- ▼ The sketch should be fully located within the basic face (i.e., it should have no common points with edges that limit the basic face).
- ▼ The sketch segments may not intersect and/or have common points.

2.4.8.4.2. Height

To define the louver height, it is necessary to select the dimension type for the louver height and enter the height value.

Three types of height dimension are available:



- ▼ **Full,**




- ▼ **From face,**





- ▼ **Slot height.**

After selection of the dimension type enter the right value in field **Height**. The range of allowed height values of a louver depends on the thickness of the sheet material (S) and the width of the louver (B). This dependency is different for different types of the height dimension (see Table 2.4.22).

Tab. 2.4.22. The range of louver height values

	Height dimension type	Height value range
	Full	$S < H < B$, where H — full louver height

Tab. 2.4.22. The range of louver height values

Height dimension type	Height value range
 From face	$S < h < B - S$, where h is the louver height above the face
 Slot height	$0.00 < A < B - 2 \cdot S$, where A is height of the louver opening

Geometrical parameters of the louver are shown in Fig. 2.4.107.

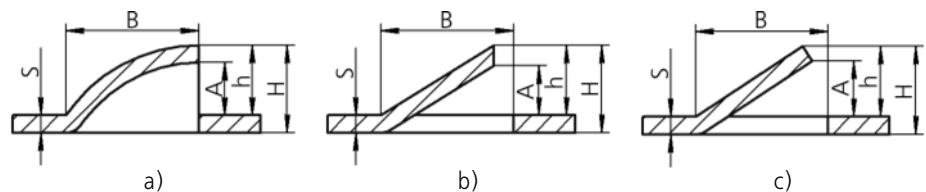


Fig. 2.4.107. Geometrical parameters of the louver: a) elongated, b) cut with the end in trimming direction, c) cut with the end along the normal line to thickness

2.4.8.4.3. Width

To set the louver width, enter the right value in field **Width**. The width value B (see Fig. 2.4.107) should meet the following condition: $B > 2 \cdot S$, where S is thickness of sheet material. For a drawn louver, the width should also satisfy the condition $B < L/2$, where L — length of a line in the louver sketch.

2.4.8.4.4. Construction Method

To set the louver building method, expand list **Method** and select the right line from it. Two methods are available: **Drawn** and **Trim**.

When using method **Trim**, you can select the end shape:



▼ **In the direction of trim** or



▼ **By thickening normal**.

2.5. Points and curves

2.5.1. Theoretical Summary

This Section contains definitions and clarifications of some terms and notions for proper understanding of this part of the Manual.

2.5.1.1. Terms and Definitions

Connected Set of Faces— a set of faces each of which has at least one common edge with at least one face of the set whereas one edge simultaneously belongs to not more than two faces.

Connected set of curves (chain of curves) is a set of curves, each of which has a common vertex at least with one curve from this set, at that, any vertex can simultaneously belong to maximum two curves.

Single-Segment Curve—curve that consists of one part. Single-Segment Curve is always selected and involved in operations entirely. Single-Segment Curves examples: an arc, a spiral, a spline.

Multi-Segment Curve— a curve that consists (or potentially may consist) of several sequentially interconnected portions— segments. Multi-Segment Curve can be selected and involved in operations and entirely (to do this, you need to specify it in the model's Design Tree), and as individual segments (to do this, they should be specified in the model window). Multi-Segment Curves examples: Polyline, contour.

2.5.1.2. Parametric representation of the curve

In general, the model file describes a **curve** as a geometrical locus of points, with spatial coordinates of these points being functions of the t parameter:

$$x = x(t),$$

$$y = y(t),$$

$$z = z(t),$$

where parameter t is limited by threshold values $t_{\min} \leq t \leq t_{\max}$.

Such curve description is its **parametric representation**.

Parametric area of the curve is the range of the t parameter values represented as a line segment (see Fig. 2.5.1, a). The curve is the image of such line segment in the 3D model. Each point on the curved line corresponds to a specific t parameter value (see Fig. 2.5.1, b).

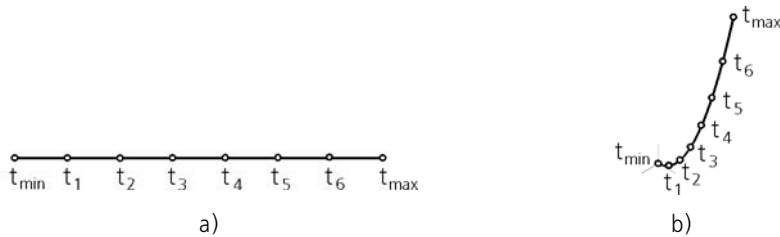


Fig. 2.5.1. Semi-cubical parabola (Neil parabola):
 a) parametric area of the curve – t parameter values;
 b) displaying t parameter values on a 3D curve



Note that generally the points evenly splitting the parametric area of the curve are displayed as points of the curve splitting it **unevenly**.

Let us consider parametric representation of the semicubic parabola (Neil parabola).

Curve Equation (or curve law):

$$y^2 = a^2 \cdot x^3, a > 0.$$

Parametric representation of the curve:

$$x = t^2,$$

$$y = a \cdot t^3,$$

$$z = 0,$$

$$-\infty < t < +\infty.$$

2.5.1.3. Parametric representation of the surface.

Isoparametric curves

The model file provides description of the **surface** as a geometrical location of points with spatial coordinates being determined by functions with reference to parameters U and V :

$$x = x(U; V),$$

$$y = y(U; V),$$

$$z = z(U; V),$$

where parameters U and V are bound by threshold values $U_{\min} \leq U \leq U_{\max}$, $V_{\min} \leq V \leq V_{\max}$.

Such surface description is its **parametric representation**.

Parametric area of the surface is the range of the U and V parameters, represented as a flat square area. The Surface is the image of such area in the 3D model. Each pair of U and V parameters' values have a related point on the surface.

If a value of either U or V parameter is fixed and another is changed, a surface curved line will be obtained. This curved line is called '**isoparametric**'. The curves produced by changing parameter U with parameter V fixed are considered isoparametric curves of direction U whereas the curves produced by changing parameter V with parameter U fixed are isoparametric curves of direction V . The curves of the both directions form the **isoparametric grid** enabling visual-

ization of the **theoretical** surface equivalent to the edge on the screen. Generally, there displayed five curves in each direction; the displayed curves correspond to 0, 25, 50, 75, 100 of U or V (Fig. 2.5.2). For example, an isoparametric grid is displayed when creating a point by the **On surface** method.

Borders of the theoretical surface may not necessarily coincide with the face contour, however, the face always lies within its theoretical surface borders. More over, unlike the face, the theoretical surface has no holes.

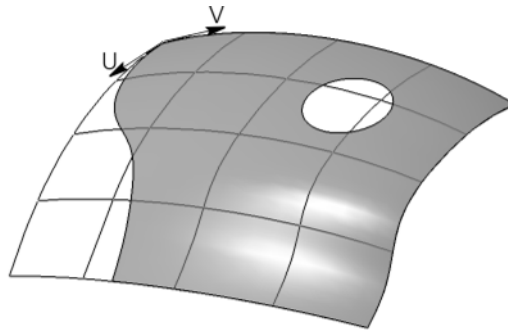


Fig. 2.5.2. Isoparametric grid example

2.5.1.4. Spline curves and surfaces Order of Curves. point weight

Spline curves and surfaces created using non-uniform rational B-splines (NURBS) are defined by the following parameters: a set of points called “**poles**” and NURBS order.

When creating a curve, custom poles can be specified. In order to create a surface, a set of poles should represent a grid built in the surface's parametric area (see Section 2.5.1.3), i.e. the poles must be positioned in rows with equal number of poles in each of them. Isoparametric curves of the future surface will run along these rows.

Evidently, the form of a spline curve or surface is defined by poles' position in space. Surfaces built by poles offer **local deformation** features that are very useful for modeling: changing position of one pole results in change of shape of only one surface part next to this point rather than the whole surface.

Spline's order in case of a curve is equal to the maximum degree of polynoms, describing sections of the curve, plus one.

The number of poles can be equal or exceed the order, but not less than the order. The order can not be less than 2. Particular cases of spline curves can be found in Fig. 2.5.3.

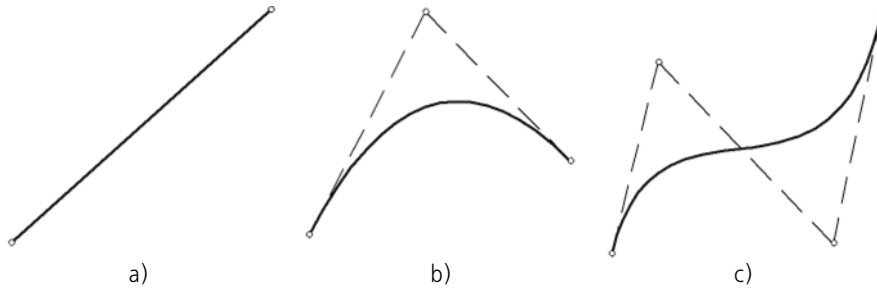


Fig. 2.5.3. Particular cases of spline curves:
 a) linear curve (number of poles – 2, order – 2, polynomial degree – 1),
 b) conic curve (number of poles – 3, order – 3, polynomial degree – 2),
 c) cubic curve (number of poles – 4, order – 4, polynomial degree – 3),

While creating a spline surface, two values defining the order by each direction should to be set: U and V. The order value set will be reflected in the isoparametric curves of the corresponding directions.



In practice, the spline order is rarely increased to anywhere larger than 6 due to significant complication (and the resulting slowing down) of calculations involved in operations with curves and surfaces.

The additional parameter which impacts on the spline curve shape is the **weight** of each pole. The geometrical meaning of this parameter is as follows: the greater is the pole's weight, the closer the curve is passing to the poles (i.e. points with a large weight “attract” the spline stronger than points with a small weight).

For example, Fig. 2.5.4 contains an example of curve shape change on increasing the weight of one of the poles. Position I of the curve corresponds to the case where all the poles have a unit weight, position II corresponds to the case where pole 4 has weight 2 whereas position III corresponds to the case when its weight equals 6.

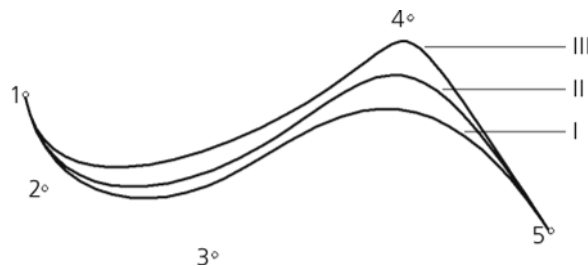


Fig. 2.5.4. Influence of pole weight on curve shape



The curve shape is defined by the difference in the absolute weight of poles and not the weights proper - i.e. the curve shape will remain unchanged on multiplying the weights of all poles by the same value.

2.5.1.5. Basis vectors at a point of the curve

Each point of the curve is associated with three mutually perpendicular planes: a normal, osculating and aligning, as well as three mutually perpendicular basis vectors: tangent vector t , vector of the principal normal n , and binormal vector b (see Fig. 2.5.5).

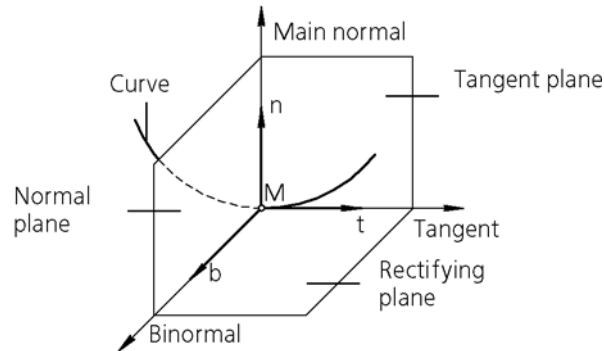


Fig. 2.5.5. Basis vectors at a point of the curve

The **tangent** and the **main normal** vectors lie in the osculating plane, and the **binormal vector is perpendicular** to this plane.

The osculating plane is the plane tending to take position of the plane passing through three curve points M , M_1 , M_2 , provided that points M_1 and M_2 tend towards M (see Fig.2.5.6).

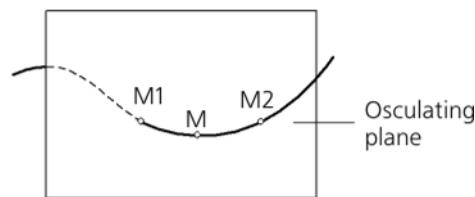


Fig. 2.5.6. Osculating plane

The plane where tangent vector and binormal vector are located is an **aligning** plane, while the plane where vector of the principal normal and binormal vector are located is a **normal** plane.

2.5.1.6. Curvature of curve Osculating circle

Curvature of the curve in point M is the value that characterizes the curve's deviation off the osculating straight line near the point in that point.

Curvature of the curve in point M can be defined using the circle tangent to it in this point.

A tangent circle is an extreme position of the circle when it goes through point M and two other points M_1 and M_2 infinitely near to it (see Fig. 2.5.7). Tangent curve lies in the **tangent** plane of the curve in point M (see Section 2.5.1.5 on p. 331 for more information about tangent plane).

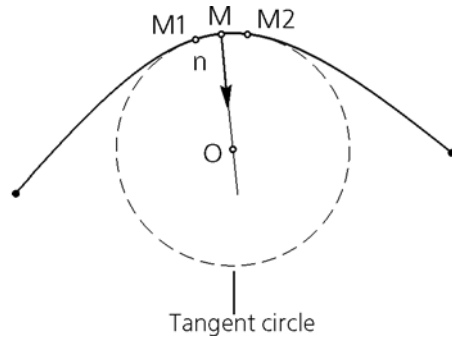


Fig. 2.5.7. Tangent circle in point M of the curve

Center of the circle tangent to the curve in point M is the **center of curvature** (O) of the curve in a given point while the radius of this circle is the **curvature R** of the curve in the point.

Radius of curvature (R) is a reciprocal of the curvature (K): $R=1/K$.



Center of the curve curvature is always located on the main normal.

2.5.1.7. Mating Condition for Curves and Surfaces


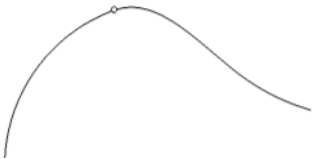



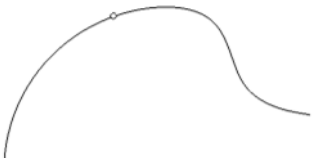
Mating condition determines the shape of curve or surface created near connection of such curve or surface to the existing curve or surface. Such existing curve or surface is a **mating object** to the created one. Curves are mated to one another and to surfaces in the **mating point** whereas surfaces are mated along a curve — the **mating border**.

Curves and surfaces mating results under various mating conditions are presented in Table 2.5.1. The Table also contains examples of curves joints under various mating conditions. The curve being the mating subject is shown on the left while the curve being mated is shown on the right. The mating point is marked with a circle.

Tab. 2.5.1. Mating condition for curves and surfaces

	Mating condition	Result of mating
	<p>Not specified The curve end point or surface boarder belong to the mating object.</p>	

Tab. 2.5.1. Mating condition for curves and surfaces

	Mating condition	Result of mating
	<p>By Tangent The Not specified condition is observed. The following additional condition is met:</p> <ul style="list-style-type: none"> ▼ for a curve: the curve and the mating subject have a common tangent at the mating point[*], ▼ for a surface: in the mating boundary points, isoparametric curves of the mated surface and of the mating surface have a common tangent. 	
	<p>Perpendicular The Not specified condition is observed. The following additional condition is met:</p> <ul style="list-style-type: none"> ▼ for a curve: the tangents of the curve and the mating subject are perpendicular at the mating point. ▼ for a surface: in the mating boundary points, tangents to isoparametric curves of the mated surface and of the mating surface are perpendicular. 	
	<p>Smooth The By Tangent condition is observed. The following additional condition is met:</p> <ul style="list-style-type: none"> ▼ for a curve: the curvature of the curve at the mating point is equal to that of the mating subject[*], ▼ for a surface: in the mating boundary points, isoparametric curves of the mated surface and of the mating surface have the same curvature. 	

* If the mating subject is represented by a surface, mating will require specification of the curve on the surface and passing through the mating point.



In some 3D modeling systems the process of mating curves or surfaces using **Not Specified** condition is referred to as “Mating with G0 Continuity”, using **By Tangent** condition - “Mating with G1 Continuity”, and using **Smooth** condition - “Mating with G2 Continuity”.

2.5.2. Points

Commands for creating 3D lines and points are located in **Operations** menu, and the buttons to call them are on the **3D Lines** panel (Fig. 2.5.8).



Fig. 2.5.8. 3D Curves bar

2.5.2.1. General procedure for point construction

Creating a dimensional point is possible using the following methods:

- ▼ **By coordinates (XYZ),**
- ▼ **By move relatively to the other point,**
- ▼ **At the intersection of two objects,**
- ▼ **On a line,**
- ▼ **On a surface,**
- ▼ **In the center of an object,**
- ▼ **By projection of a point on an object,**
- ▼ **By cylinder (RAZ),**
- ▼ **By sphere (RAB).**

The use of the specified object in each method provides an associative relation of the point to this object, for instance, pertaining of a point to a surface.

To create a dimensional point, apply the command **Point**.

Select a variant of point construction from the **Method** list or using commands of the context menu. The elements for managing construction will appear on the Properties Bar (Fig. 2.5.9).

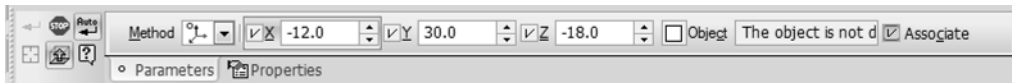


Fig. 2.5.9. Property Bar for point construction by coordinates

Then specify parameters and objects used for construction.

Along with input of numerical values you can move the characteristic point by mouse to define the final position of the point, e.g. while creating on a curve. In such a case the relation with the object won't fail and the parameters shall be calculated and displayed in the fields of the Property Bar.

Parameter input for each method is detailed in Sections 2.5.2.2 — 2.5.2.9.

The ghost point appears in the model when all the required data has been specified. While construction is not yet finished, the location of the point can be changed by activation of the required control element on the Property Bar.

Before completion of the command the system "remembers" the objects specified in each construction method, therefore you can switch between the methods without loss of the objects. The exclusions are **By Coordinates (XYZ)**, **By Cylinder (RAZ)**, **By sphere (RAB)** methods: switching to any of them will result in automatic creation of the point that coincides with the point defined in the last of the selected methods. Due to it, at any moment of point construction

you can view its current coordinates in the Cartesian, cylindrical or spherical coordinate system.



To create a point without an associative link with the objects used during construction, switch, for example, to the **By Coordinates (XYZ)** method and complete the command.



To change the objects used for point construction by means of the selected method, click the **Select Other** button on Special Control Panel and repeat object selection.

Name, color and display style of a point can be customized on the **Properties** page of the Property bar.



To finalize the construction, click the **Create Object** button on the Special Control Panel.



In the **By Coordinates** and **In Center** methods, automatic creation of objects is available and enabled by default.

2.5.2.1.1. Objects Used for Construction

During construction, two types of objects are used: objects that include the point and auxiliary objects: base points and guide objects (in **Move** and **Projection** methods).

Points in the model can be located on spatial curves and surfaces and also can coincide with point objects.

Point Objects are:

- ▼ Coordinate system origins,
- ▼ sketch line vertices,
- ▼ Standalone points in sketches,
- ▼ standalone dimensional points,
- ▼ vertices of dimensional curves,
- ▼ edge vertices.

Base Point is the reference point for offset in the **Move** method and the projected object in the **Projection** method. Any point object can be a base point.

Guide Objects designated to set the offset direction or the point projection can be the following objects of a model.

- ▼ Point objects (listed above).

The direction defined by a point object is a line passing through this point object and the specified base point.

- ▼ Straight objects:
 - ▼ sketch line segments,
 - ▼ polyline segments,
 - ▼ coordinate and auxiliary axes,
 - ▼ straight edges.

Direction, defined by a straight object — a line parallel to the object.

- ▼ Flat (planar) objects
 - ▼ coordinate and auxiliary planes,
 - ▼ flat faces.

Direction, defined by a plane object — a line perpendicular to the object.



Moreover, the guide object may be represented by a vector. This vector can be created by means of **Vector drawing** command (see Section 2.8.5 on p. 613).

2.5.2.2. Construction of point by coordinates XYZ



By Coordinates (XYZ) is a method of point construction allowing you to directly set its Cartesian coordinates in space or link it to an existing point object.

Point coordinates can be input from a keyboard or defined automatically by positioning the mouse in the model window. Associative linking of the point with the point object is enabled and disabled by the **Associate** option.

- ▼ If all the three point coordinates are known, enter them to the corresponding fields of the Property Bar and fix them by pressing the <Enter> key.
- ▼ When you move the mouse in the model window, the point is defined as the cursor projection on the surface parallel to screen and running through the origin of coordinates. Current coordinates of the point are displayed in the **X**, **Y**, **Z** fields of the Property Bar. When you move the cursor to the point object (list of objects see in the Section 2.5.2.1.1), a cosmetic vertex or point appears next to it, and the coordinates are defined as those coinciding with the coordinates of this point object. To fix the point with the current coordinates, click the left button of the mouse.

Selecting of the point object when **Associate** option is on will enable the forming of associative relation between the point and this object.



The presence of relation is manifested by displaying a checkmark on the **Object** button and the name of the point object in the **Object** field on the Property Bar. Due to this relation, a newly created point will follow the object when the latter is repositioned.



To cancel the associative relation with the point object, click the **Object** button. The tick on the button will disappear.



To restore an associative relation of a point and a point object, enable the **Auto association** option and specify the required object anew.

- ▼ If two coordinates of the point are known, enter them into the corresponding fields of the Property Bar and fix them. While you move the mouse, the point will move along the straight line defined by the specified coordinates. To create a point as a projection of any point object on this line, disable the **Associate** option and select the required point object. To fix the point, click the left button of the mouse.
- ▼ If one coordinate of the point is known, enter it into the corresponding field of the Property Bar and fix it. While moving the mouse, the point will move along the plane defined by the specified coordinate. To create a point as a projection of any project object on this plane, disable the **As-**

sociate option and select the required point object. To fix the point, click the left button of the mouse.



Point coordinates are specified in that point's coordinate system (for object coordinate system refer to Section 2.8.3.4 on p. 597).

2.5.2.3. Moving a Point with respect to Object



Move is a method of point construction by defining the coordinates in relation to the base point or by offsetting from it for the predefined distance in the specified direction.

For base points, you can use the same point objects as those used for linking the points to the object in the **By Coordinates** method.

To create a point by the **Move** method, select a base point, i. e. the point specified as the coordinate origin or the reference point to count the offset distance.



To select the base point, specify the required object while clicking the **Base Point** button. After a cosmetic vertex or a point appears next to the cursor, click the left mouse button. Its name will appear in the **Base Point** field.

2.5.2.3.1. Move by coordinates

Fields for coordinates input become accessible after specifying the base point.

Enter the relative coordinates of the point being created in the **X,Y,Z** fields of the Property bar and fix by pressing the *<Enter>* key.



Relative coordinates are set in the Cartesian coordinate system, the origin of which is a base point and the axes run in the same directions as the axes of coordinate system of the point being created (for object coordinate system see Section 2.8.3.4 on p. 597).

2.5.2.3.2. Move Relatively to Guide Object



After specifying the base point, press the **Guide Object** button.

In the model, select the object to define the direction for the offset of the point to be created from the base point. The list of guide objects and directions they define are specified in Section 2.5.2.1.1 on p. 335.



Moreover, the guide object may be represented by a vector. Such vector can be created by means of **Vector drawing** command (see Section 2.8.5 on p. 613).

In the **Guide Object** field, the name of the selected object will appear. If the direction is set by means of a vector, the field will contain the word "Vector".

After the guide object is specified, the **Distance** field becomes accessible.

Specify the distance from the base point to the point to be created by entering it into this field or selecting the point's position by mouse in the model window.



Instead of the distance to be defined, you can specify a point object. In this case, a point will be created that is a projection of the specified object onto the offset direction.

2.5.2.4. Point construction on Intersection of Objects



Intersection is a method to construct a point on the intersection of two objects. Objects are axes, dimensional curves and their segments, edges, sketch lines, faces of surfaces and solids. The point is automatically created subject to the required condition: availability of a finite number of object intersection points.

To create a point, complete the following steps.



1. While the **Object 1** button is pressed, select by cursor an object in the model: a curve or a surface. The name of the object selected shall be displayed in the field **Object 1**.



2. While the **Object 2** button is pressed, select by cursor the second object in the model that intersects the first object. The name of the object selected shall be displayed in the field **Object 2**. If the object does not have any common points with the first one or the number of common points is infinite, then the definition in the **Object 2** field will be unavailable. In this case, you should specify the second object again or select another pair of objects.

You can select objects in any sequence.

After both objects are defined, a ghost of the point will appear on the screen.

To use another object for performing the command, click the **Object 1** or the **Object 2** button. Selection of an object will be cancelled and you will be able to select a new object.



If there are several points of intersection, then they will be displayed one-by-one when you click the **Next Object** buttons and **Previous Object** on the Special Control Panel. After the required point appears on the screen, finish the construction of the object.



2.5.2.5. Point Construction on Curve



On Curve is a method to create a point offset along the curve for some distance from its vertex. The distance can be specified in relative or absolute values.

You can select a 3D curve or its segment, a sketch line or an edge as a curve.

To create a point on a curve, perform the following steps.

1. Select a curve in the model by cursor. The selected curve will be highlighted in the model. Two ghosts will be created on the curve: a point in the specified place and a start vertex with a directing arrow to count the distance. At the same time, in the **Curve** field on the Property Bar, the name of the selected object will be displayed.
2. Move the characteristic point by mouse along the curve to the required position.

Also, you can specify the offset numerical value on the Property Bar. To do this select a specification option from the **Offset** drop-down list containing the available method options.



- ▼ **At % of curve length** by relative value of the offset from the initial point to the characteristic point. The percentage value for entering into the **% of curve length** input field should be from 0 to 100 of the curve length.



- ▼ **By Segment Length**— by the length of the curve from the initial point to the characteristic one. The **Segment Length** field is used to specify the absolute offset in the length measurement units.



- ▼ **By Arc Central Angle** — by the central angle between a initial point to the characteristic one. The variant is available when an arc of circle or ellipse is selected as a curve. The **Angle** field is used to set the central angle of the arc in angle measurement units.

After the curve is selected, the field for the current parameter will display its value in the point on the curve. When moving the point by mouse, the parameters are calculated and put to these fields.

Enter or specify by counter the value of the offset parameter.



During construction, you can switch between the options. In this case the offset value will be calculated and displayed in units of the selected option.

You can construct a point on a curve by linking it with an existing point object. For that, select a point object (for list of objects — see Section 2.5.2.1.1 on p. 335).

The position of the point is determined by the shortest distance from point object to the curve.



Owing to associative link, the newly created point will follow the object whenever its position is changed. The presence of relation is manifested by displaying a checkmark on the **Object** button and the name of the point object in the **Object** field on the Properties Bar. In this case the field for inputting point offset along the curve is not available for editing.



To cancel the associative relation with the point object, click the **Object** button. The tick on the button will disappear. After that, the point offset field is editable again.



To restore the associative relation of the point with the point object, specify the required object again.



While the **From Start Vertex** button is pressed of the **Direction** switch group, the offset is measured from the start vertex of the curve in the direction of the offset vector. To change direction, click the **From End Vertex** button.



Unless the offset is not specified by the user, the change of direction will not change the point position on the curve.

2.5.2.6. Construction on Surface



On surface is a method to construct a point lying on a surface. The following can be used as surfaces:

- ▼ faces of surfaces and bodies,
- ▼ coordinate planes.

The point's position can be set:

- ▼ by moving theoretical surface of the selected face along isoparametric curves U and V,
- ▼ by moving (linear or angular) the point from two specified reference objects;
- ▼ coordinates on the surfaces:
 - ▼ flat (two types of coordinates—orthogonal and polar—are available),
 - ▼ cylindrical (in cylindrical coordinate system),
 - ▼ spherical (in spherical coordinate system).

For more information on theoretical face's surface and isoparametric curves, see Section 2.5.1.3 on p. 328.

In order to create a point on a surface, it is necessary to specify the surface, choose the variant of point position specification, and specify parameters that define the position.



In order to specify the surface, use the cursor to select a face or plane in the model while the **Surface** button is enabled. The selected object will be highlighted and its name will appear in the **Surface** field.

In order to choose the variant of specifying the point's position, choose the respective line in the **Offset** list. Variants are described in detail in Sections 2.5.2.6.1-2.5.2.6.4.

The **Allow for borders** option makes it possible to limit the point's position by the face's contour: if the option is enabled, the point may be positioned inside the contour; if the option is disabled, it can be positioned both inside and outside (if necessary, the face's theoretic surface will be extended). If you build the point on an auxiliary or coordinate plane, the **Allow for Borders** option will not be available.

2.5.2.6.1. By parameters U and V



This variant of specifying the point's position is used when the relative values are known or if you need to create a point lying on a theoretical surface.

If this variant is chosen, the ghost of the face's theoretical surface appears on it as a grid of intersected isoparametric curves. The ghost contains the following indications:

- ▼ arrows to show the directions of U and V parameters effect in the start point $U=0, V=0$;
- ▼ dashed lines to indicate ghosts of U and V parameters in the specified point.

In the Property bar, the following fields are present: **Parameter U, %** and **Parameter V, %**. In these fields, the current parameters U and V in the point of face specification will be displayed.

Methods of specifying the position of a point on a surface

▼ Moving the handle in the model window

When moving a handle (about handles, see Section 2.1.4.2.2) current parameters setting the position of the point are displayed in respective fields.

▼ Inputting parameters setting the position of a point in respective fields on the Properties bar

Input U and V parameters or set them using counter and fix by pressing *<Enter>*.

▼ Linking with point object

Move the pointer to the point object (list of objects — see Section 2.5.2.1.1 on p. 335) — a symbol of vertex or point will appear next to the object. The point's position is defined as a projection of the specified point object on the construction surface. Owing to associative link, the newly created point will follow the object whenever its position is changed.



The presence of relation is manifested by displaying a checkmark on the **Object** button and the name of the point object in the **Object** field on the Properties bar. U and V parameters are displayed in respective fields on the Properties bar, and they are not editable.



To cancel the associative relation with the point object, click the **Object** button. The tick on the button will disappear.



To restore the associative relation of the point with the point object, specify the required object again.

2.5.2.6.2. By offsets from two objects



The **By offsets from two objects** variant is applied if offset (linear or angular) of the point from two reference objects is known.

In the **general case**, the point is located on an arbitrary surface, while offset is set by distances between it and two flat objects. The following objects may be used as flat objects:

- ▼ flat faces,
- ▼ auxiliary and coordinate surfaces,
- ▼ edges in the form of flat curves,
- ▼ sketches (in the Design Tree).

Flat objects must meet the following requirements:

- ▼ they must be non-parallel to each other,
- ▼ the line of intersection of their planes should not be parallel to the surface (or forming surface) on which the point is located.

Besides that, the following **special cases** are possible:

- ▼ if the point is positioned on a *flat face*, you can use a straight-line edge as a reference object for it; in this case, offset will be counted from the plane going through the edge perpendicular to the flat face;
- ▼ if the point is positioned on a *rotation surface (except for spherical surface)*, you can specify an angular offset from the plane going through the surface's axis (offset from the second object will be linear in this case); in order to specify such plane, you can specify a coincident flat object and a straight-line edge within it.



If the **By offsets from two objects** variant is selected, the **Object 1** and **Object 2** fields will appear in the Property bar. These fields will display names of objects from which the point's offset is calculated. The objects are automatically identified by the system.



Groups of switches **Type of Parameter 1** and **Type of Parameter 2** can be used for specifying the offset type (linear or angular).



You can switch between parameter types, if the point is built on the rotation surface, and a plane going through the rotation axis is selected as a reference object. In other cases, the parameter type is defined by the system automatically and it is impossible to change it.

The **Distance 1/Angle 1** and **Distance 2/Angle 2** fields display calculated offsets of the point at its current position with respect to specified objects.

The model window displays arrows denoting positive direction of offset counting.

Edit offset values; if necessary, change reference objects and/or offset type. Note that entry of offsets is possible only if both reference objects are defined.



As the second reference object, you can specify only such an object which results in finite number of points when surface and the first reference objects are selected.

Methods of setting the position of a point are similar to those used in the **By U and V parameters** (see Section 2.5.2.6.1 on p. 340).



If several points on the surface meet the defined conditions, the following buttons will become available on the Special Control panel: **Next object** and **Previous object**. Using these buttons, choose a point for construction and finish creating the object.

2.5.2.6.3. By coordinates on plane



The **By coordinates on plane** variant is applied if you need to create a point on plane by specifying its rectangular or polar coordinates.

If you use this variant, the Property bar will display the **Type of coordinates** group of switches for selecting the type of coordinates when constructing the point:



▼ **Rectangular coordinates,**



▼ **Polar coordinates.**

If **rectangular coordinates** are chosen for creation of the point, **X** and **Y** fields will appear in the Property bar; if **polar coordinates** are chosen, **R** and **A** fields will appear in the Property bar. By default, the point is located in the point of plane specification. Current coordinates of the point are displayed in the respective fields of the Property Bar.

Methods of setting the position of a point are similar to those used in the **By U and V parameters** (see Section 2.5.2.6.1 on p. 340).

2.5.2.6.4. By coordinates on cylinder and sphere



The **By coordinates on cylinder** variant is used if you need to construct a point on a cylindrical surface using **cylindrical coordinates**. When you choose the method, the **A** and **Z** fields appear on the Properties bar.



The **By coordinates on sphere** variant is used if you need to construct a point on a spherical surface using **spherical coordinates**. When you choose the method, the **A** and **B** fields appear on the Properties bar.

For more details on cylindrical and spherical coordinates, see Section 2.5.3.4.1 on p. 358.

To construct a point in cylindrical coordinates it is required that the origin of the current coordinate system is on the axis of a cylindrical surface that coincides with the Z axis of the coordinate system, while to construct a point in spherical coordinates it is required that the origin of the current coordinate system is the center of a sphere.

Choosing the **By coordinates on cylinder (sphere)** variant is available, if there is a cylindrical (spherical) surface meeting the above requirements on which the point is to be placed. An vice versa, if the variant is selected, you can select only such cylindrical (spherical) surfaces that meet the requirements.

By default, the point is placed in the point specified on the surface. Current coordinates of the point are displayed in the respective fields of the Property Bar.

Methods of setting the position of a point are similar to those used in the **By U and V parameters** variant (see Section 2.5.2.6.1 on p. 340).

2.5.2.7. Construction of point in the Center of Object



In Center is a method to create a point in the center of an object (for bodies and faces — in the center of gravity).

The objects are:

- ▼ sketch contours being circles, ellipses or their arcs,
- ▼ sketch lines being circles, ellipses or their arcs,
- ▼ spherical surfaces,
- ▼ bodies,
- ▼ faces.

To create a point, select an object. The object's name will be displayed in the **Object** field. In the model, the point will be created automatically.



Bodies are selected in the Design Tree of the model. The body icons and names are shown on the Tree if the model structure is enabled (see Section 2.1.2.2 on p. 106).



When calculating the face's mass center, only its external contour is taken into account.

2.5.2.8. Construction of Projection of Point on Object



Projection is a method to create a point as a projection of a base point on an object.

Any point object can be a base point.

Objects on which a base point is projected are:

- ▼ faces, auxiliary and coordinate surfaces;
- ▼ edges, dimensional curves, sketch lines, auxiliary straights, coordinate axes.

Projection can be orthogonal or constructed in the direction specified by an object. The list of objects and directions they determine are indicated in Section 2.5.2.1.1 on p. 335.

Moreover, the guide object may be represented by a vector. Such vector can be created by means of **Vector drawing** command (see Section 2.8.5 on p. 613).



To create a projection, complete the following steps.



1. Select a base point. The name of the point will be displayed in **Base Point** field.



2. With the **Object** button pressed, select the object, which the base point is projected on. The name of the object will be displayed in the **Object** field, and in the model, a ghost of an orthogonal projection of the point will appear.

3. Finish creation of the point, if construction of orthogonal projection is required.



Select the guide object or create a vector in order to construct a projection in a pre-determined direction. Click button **Guide Object** and then select the direction reference object or create a vector.

The ghost will change its location. Complete the point creation.



The point is built only if the projection falls into the object and not into its continuation. Exceptions are the flat surface and revolution surface— projection may be created on their extensions.

Auxiliary and coordinate planes and axes are infinite, therefore, a projection is created regardless of its position respect to the rectangle or the line segment, representing a flat or an axis.

2.5.2.9. Construction by Cylindrical (RAZ) and Spherical (RAB) Coordinates



By Cylinder (RAZ) is a method of point construction that allows you to set the point's cylindrical coordinates (R, A, Z) in space or link it to an existing point object.



By Sphere (RAB) is a point construction method allowing you to set the point's spherical coordinates (R, A, B) in space or link it to an existing point object.

For more information about the cylindrical and spherical coordinate systems location relatively to the coordinate system see Section 2.5.3.4.1 on p. 358.

You can set sphere or cylinder radius by entering its value in the R coordinate field or by selecting an object defining a radius value.



To select the object defining radius, enable the **Radius Object** and select the required object:

- ▼ an edge or a line in a sketch having the form of a circle or a circle arc,
- ▼ a curve — a circle or an arc,
- ▼ a cylindrical or spherical edge.

The value of the specified object's radius is specified in the **R** field. An associative relation is formed between the point and the object. Therefore, the **R** field becomes unavailable. To cancel the link, enable the **Radius Object** switch and reselect the object.

Other methods used for point creation **By Cylinder (RAZ)** and **By Sphere (RAB)** i.e. input and locking of point coordinates, linking a point to the already existing point object, are the same as those used for point creation **By Coordinates (XYZ)** (see Section 2.5.2.2 on p. 336).



If two objects are selected for point creation: point radius and defining radius, then a point is created as a projection of a point object on a cylinder or a sphere of respective radius.

2.5.2.10. Point Variables

After a point is created in a model, the variables of this point are automatically created.

The point variables include three informational ones. They correspond to the Cartesian coordinates of a point. These variables are always created.

Availability and contents of other variables are preconditioned by the point creation method (Table 2.5.2).

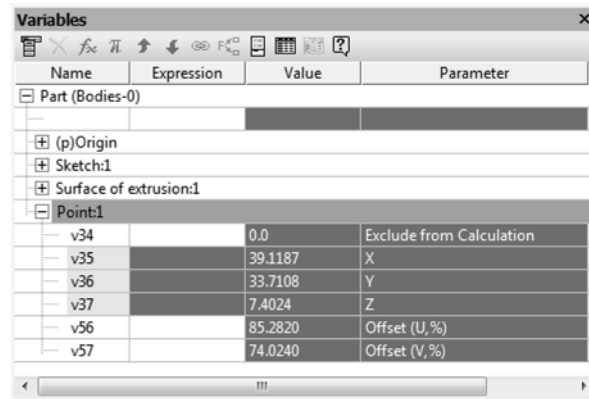
Tab. 2.5.2. Point Variables

Point construction method	Variables
By Coordinates	<ul style="list-style-type: none"> ▼ X coordinate ▼ Y coordinate ▼ Z coordinate
Move	<ul style="list-style-type: none"> ▼ relative coordinate X ▼ relative coordinate Y ▼ relative coordinate Z or ▼ offset distance
On curve	<ul style="list-style-type: none"> ▼ offset value (offset amount in %, the arc length, or central angle) ▼ offset direction
On surface	<ul style="list-style-type: none"> ▼ U parameter value in % ▼ V parameter value in % or ▼ offset from the first flat object ▼ offset from the second flat object or ▼ X (or R) coordinate ▼ Y (or A) coordinate



If the point is created in local system of coordinates (rather than in absolute system), most construction methods will result in such point having only informational variables.

Figure 2.5.10 illustrates the variables of the point created by the **On Surface** method.



Name	Expression	Value	Parameter
Part (Bodies-0)			
(p)Origin			
Sketch:1			
Surface of extrusion:1			
Point:1			
v34		0.0	Exclude from Calculation
v35		39.1187	X
v36		33.7108	Y
v37		7.4024	Z
v56		85.2820	Offset (U,%)
v57		74.0240	Offset (V,%)

Fig. 2.5.10. Point variables

Work with variables and expressions is described in detail in Section 7.1 on p. 1569.

2.5.2.11. Using point command functionality while creating other objects

To construct a point using options available in the **Point** command is possible by the following commands:

- ▼ Spline,
- ▼ Polyline,
- ▼ Line segment,
- ▼ Graphic Circle Arc,
- ▼ Surface created by the point grid,
- ▼ Surface on grid,
- ▼ Control point,
- ▼ Attachment Point,
- ▼ Axis through the Vertex,
- ▼ Scaling;
- ▼ Plane Tangential to Face at Point;
- ▼ Plane through Vertex Perpendicular to Edge,
- ▼ Create zone,
- ▼ Create/edit section,
- ▼ Add from file,
- ▼ Add local part from file,
- ▼ Base part,
- ▼ Place component.

The steps for forming a contour through tracing by arrow are described in Section 2.5.2.11.1.

It should be noted that execution of the **Axis Through Vertex Along Object** and **Scaling** commands as well as those enabling construction of auxiliary planes results in visualization of constructed points as individual objects in the Design Tree. Editing these points result in changing their parameters.

The points constructed using other commands are stored inside the objects created by such commands and are not displayed in the Design Tree. You can change their parameters directly rather than through editing the objects such points were created for. For information about the procedure and options of editing such points, see Section 2.5.2.11.2.

When adding a component to a model or editing its position parameters, and when creating/editing model sections new points are not created at all. The point construction process is used only for determining the position of the point of inserting a component or the points that determine the position of a section (flat or block-like).

2.5.2.11.1. Constructing a point during object creation



To finalize the construction, click the **Create Point** button on the Special Control Panel.

System starts a point creation process fully corresponding to the process enabled by the command **Point** (see Section 2.5.2.1 on p. 334). Select a point creation type, then subsequently specify the required parameters (enter the values or specify the objects). The created point is displayed as a ghost.



Upon completion of the point parameterization click the **Create Object** button on Special Control Panel.

The systems gets back to the command for which the point was constructed.

Name of the point construction type is highlighted in the special field on the Property Bar. If the point is constructed **By coordinates (XYZ)**, the point coordinates will be highlighted.

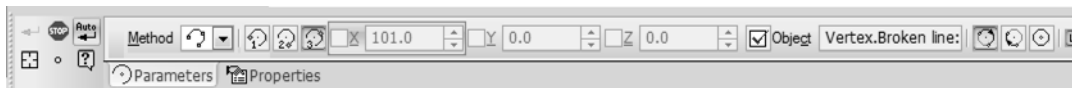


Fig. 2.5.11. Parameters of the point built while constructing an arc displayed in the Properties bar



Please note the following peculiarities of constructing points when creating a polyline or spline.

- ▼ When constructing a point while button **Create Object** is pressed, the point becomes fixed with no return to polyline or spline construction process. This allows to create several curve vertices in sequence. To finish the point creation process, click the **Create Object** button.
- ▼ The spline or polyline parameters table will highlight the point coordinates without their construction types.

2.5.2.11.2. Edit a point constructed during object creation

1. Enter the mode to edit the inscription.
 2. Select (in the model window or with the aid of the Property Bar control elements) an object point to be edited. No selection is necessary while editing a control or attachment point.
- Note: When you move cursor to the point, its construction method and coordinates will be dis-

played next to the cursor. For control or attachment point, the method of point construction is displayed in the **Point** information field on the Property bar.

3. Set the new position of the point by doing one of the following:

- ▼ In the model window select the point object. Information about the constructed point is erased, and the edited point coincides with the specified object. If option **Associate** is enabled on the Property bar, the point will be associated with the object.



- ▼ Click the **Create Object** button on the Special Control Panel. The object editing process starts, which is similar to the initial construction process. Change parameters of the active point creation type or select another type and create a new point. After finishing editing, click the **Create Object** button to exit point edition process.

- ▼ When editing an arc, surface, spline or polylines a point can be moved with a mouse in accordance with the point construction method. For example, the point on curve can be moved along this curve. Some construction methods, e.g., **At Intersection**, do not allow for the point free movements.

- ▼ A point associated with a point object can not be moved in arbitrary manner — it follows the point object. You can delete relation. To do this, complete the following steps:



- ▼ in the arc edit mode: click the **Object** button on the Property bar,

- ▼ in the spline or polyline edit mode: disable option in the **Relation with Point Object** column in the Table of vertices,



- ▼ in the surface edit mode: enable the point edit mode by clicking **Create Point** on the Special Control panel, click **Object** on the Property bar, and then — **Create Object**.



Upon deletion of the associative relation, the point remains coincided with the point, however, it can be freely moved with a mouse.



Control or attachment point is always object-associated.

4. After finishing editing, click the **Create Object** button. The sketch will be highlighted in the model window.

2.5.3. Point Groups

Spatial points can be created in groups on an existing object — i.e. a curve or a surface. Such groups can be created using the points coordinates saved in a file.

Commands to create point groups are located in the **Operations** menu.

Their shortcuts are grouped on the **3D Curves** panel (Fig. 2.5.12).



Fig. 2.5.12. Commands to Create Point Groups

2.5.3.1. General Techniques for Creating Point Groups

After calling the group creation command select the object whose surface would contain the group elements. Its name will appear in the field on the Properties bar.

If the point group is created using the data from a file, select the file name and path. The file name will appear on the Properties bar.

Set the parameters of the group and the point properties using the fields and switches on the Properties Bar.

The ghost of the point group is displayed on the screen. This allows you to evaluate correctness of parameterization and selection of initial features.

If necessary, change the group's name, color, and point display style. For that, use control elements on the **Properties** tab of the Property bar.



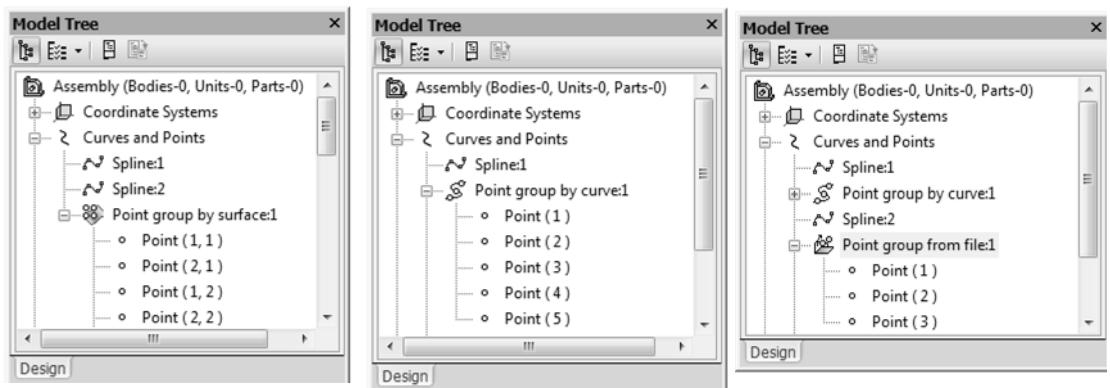
Confirm group creation.

The created group of points will appear in the model window, and the icon corresponding to its type — in the Design Tree (Fig. 2.5.13 on p. 349).

A point group consists of **elements**.

The point group elements are displayed in the Design Tree as separate objects subordinated to the group.

To expand the list of elements, click the mouse on the “plus” sign located at the left of the group icon in the Design Tree.



a) Along surface;

b) Along curve;

c) From file.

Fig. 2.5.13. Point group in the Design Tree

The group element ID number is displayed in parentheses to the right of the group element icon.

If the group is built along a surface, the group element number will contain two numbers. The first value is the element number by V parameter whereas the second one is the element number by U parameter. Instance numbering begins with one.

A point group can be broken up into separate elements (see Section 2.5.3.1.1 on p. 350).

If the **Scatter** option was selected while creating the group, the position of points in completing the command will correspond to the settings. However, they will not be the group elements. The status of the resulting points will be identical to the result of the **Explode** command.

2.5.3.1.1. Exploding a group of points

A point group can be broken up into separate elements.

You can do this by selecting a group of points to be exploded in the Design Tree and using the **Explode** command from the **Editor** menu.



In order to explode a single group of points, the **Explode** command may be called from the point group's context menu in the Design Tree.

Upon calling the command, an explode dialog will appear. To confirm explosion, click **OK** in this dialog.

The rules of exploding groups are as follows:

- ▼ icon of the group and of its objects will be deleted from the Design Tree;
- ▼ objects that used to constitute the group retain their positions in the group.

The types of objects generated by array explosion depend on the types of the group's source objects (Table 2.5.3).

Tab. 2.5.3. Results of Exploding a Group of Objects

Source Object Type	Types of Objects Resulting from Group Explosion
Group of Points Along a Curve	Point created by the On Curve method
Point groups along a surface	Point created by the On Surface method
Group of points from a file	A point created using By coordinates (XYZ) , By cylinder (RAZ) or By sphere (RAB) methods depending on the interpretation method with respect to data obtained from the file.
group of isoparametric curves on the surface,	Isoparametric curve.

2.5.3.2. Point Group Along a Curve



To create a point group along a curve run the **Point Group Along Curve** command.

Select the curve to contain the points. You can select a 3D curve or its segment, a sketch line or an edge as a curve.

The object name will appear in the **Curve** field.

The group ghost will appear in the model. The end points of the ghost are its handles. The handles can be used for setting the **working segment of the curve** — i.e. the part of the curve within which the point group is to be constructed.

The number of points in the group and their relative position can be managed by means of the controls which become accessible on selection of one of the options from the **Method** drop-down list (see Section 2.5.3.2.1).

The position of the working section of the curve can be controlled the following ways:

- ▼ By moving the handles with the mouse;
- ▼ By using the controls located in the Properties bar (see Section 2.5.3.2.2).

2.5.3.2.1. Setting the position of points along the working segment

The methods of positioning the group points along the working segment of the curve can be set selected from the **Method** drop-down list on the **Parameters** tab of the Properties bar.

Method Uniformly Along Length



Group of points creation with the use of this method is processed as follows.

1. The start and end points of the group coincided with the respective boundaries of the working segment of the curve.
2. The remaining points are positioned between them at an equal step along the curve.

Input or select in the counter desired number of group points into the **Number of Points** field.

Method Step Along the Curve



Group of points creation with the use of this method is processed as follows.

1. The start point of the group coincided with the start boundary of the working segment of the curve.
2. The remaining points are positioned after it with preset step along the curve.

Input or select in the counter required distance between points in the group into the **Pitch** field.

The **Along Entire Curve** option is available for this method. If it is enabled, within the whole working section of the curve there will be created the group with the number of points equal to the result of integer division of the curve length by the indicated step plus one point. This result will appear in the **Number of Points** read-only field.

If the option is disabled, the **Number of Points** field becomes editable. From the starting point of the working section there will be created as many points with the selected step as has been set in this field.



The product of the step by the number of points less one must not exceed the curve length.

Method

Uniform Step by Curve Parameter



Group of points creation with the use of this method is processed as follows.

1. The start and end points of the group coincided with the respective boundaries of the working segment of the curve.
2. There are points in the parametric field of the curve (see Section 2.5.1.2 on p. 327) related to the boundaries of the working segment of the curve. A part of the parametric field limited by these points is split into equal parts. Number of parts is equal to $n-1$, where n is the assigned number of points.
3. Splitting points are reflected to the curve from the parametric field. The remaining points of the group will be aligned with the points found.

Use the counter to select or type the number of points to be created in the **Number of Points** field.

If the curve on which the points must be situated is **Multi-Segment**, group of points creation is processed as follows.

1. Every curve segment has its parametric area. Parametric areas of the curve segments are equal to each other and contain the **total area** within which a group of points is created.
2. The start and end points of the group are mated with the respective boundaries of the total area.
3. Total area is split into equal sectors (see Fig. 2.5.14, a). Number of parts is equal to $n-1$, where n is the assigned number of points.
4. Splitting points are reflected from the total area to the curve's segments (see Fig. 2.5.14, b). The remaining points of the group will be aligned with the points found.

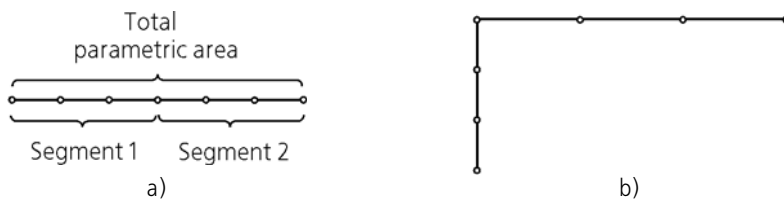


Fig. 2.5.14. Polyline as an example of creating a group of 7 points:
 a) splitting of the total parametric field of the polyline into 6 equal segments,
 b) reflection of the splitting points on the polyline's segments

2.5.3.2.2. Setting the position of working segment on the curve

The controls enabling setting of parameters which determine the position of working segment on the curve are available for all the construction methods.



The **Change Direction** switch enables selection of the curve vertex to serve as the starting point of the working segment.

By default groups are constructed from the end points of a curve. The **Offsets** option enables setting the working segment offset from the curve vertices. If this option is activated, the user may access the controls for the offsets. The option is enabled automatically if you reposition the working section by moving handles.



Group of switches **Working Segment Border** enables setting the extreme point of the working segment for which the offsets are being determined. You can set offsets for the starting and end points by using these switches one after another.

Also the user may specify the numerical values for each offset on the Properties bar. To do this, you must select a specification option from the **Offset** drop-down list containing the parameter types.



- ▼ **In % of curve length.** The percentage value for entering into the **% of curve length** input field should be from 0 to 100 of the curve length.



- ▼ **By segment length.** The **Segment Length** field is used to specify the absolute offset in the length measurement units.



- ▼ **By Central Angle of Arc.** The **Angle** field is used to set the central angle of the arc in angle measurement units. The method is available when an arc of a circle or an ellipse is selected as a curve.

The parameters are calculated and put into these fields while the handle is being dragged with the mouse.

Set the offset parameter value by using the counter or the keyboard.



During construction, you can switch between the options. In this case the offset value will be calculated and displayed in the units of the selected option.



With the **From Start Vertex** button of the **Direction** switch group pressed, the offset is measured from the start vertex of the curve in the direction of the offset vector. To change direction, click the **From End Vertex** button.

2.5.3.3. Point Group Along Curve

You can create a group of points on a surface. The surfaces may be faces of surfaces and solids.

The command execution results in a pattern of the grid of points. The points of the grid can be represented by the vertices of a polyhedron approximating the initial surface based on the specified parameters.

In order to create a group of points on a surface, do the following.



1. Call the **Point Group on Surface** command.
2. After calling the command, the **Surface** switch is enabled on the Property bar automatically. Specify the face to which the points should belong.



The object selection is detailed in Section 2.1.4.1.1 on p. 130.

The name of the object will be displayed in the **Surface** field of the Properties bar upon selection of the object.





The selected face will be highlighted in the model. The following objects will appear on the face:

- ▼ ghost of its theoretical surface; arrows to show the directions of U and V parameters effect in the start point $U=0, V=0$;
- ▼ ghost of a point group; dashed lines to indicate ghosts of U and V parameters in the specified point.

For more information on theoretical face surface, see Section 2.5.1.3 on p. 328.

3. Choose the method of points' location within the group from the **Method** drop-down list in the Property Bar. Then, set parameters required for this method. The methods for creating groups are given in Table 2.5.4.

Tab. 2.5.4. Methods of constructing point groups along a surface

Method	Construction procedure
 By the Number of Points along U and V Axes	<p>The number of points in U and V directions is user-defined. Use the counter to select or type the number of points in this direction in the Number of points, U and Number of points, V fields.</p>
 By Linear Deviation*	<p>The points are constructed in such a way that the deflection of edges of the approximating polyhedron connecting the adjacent points in U and V directions from the surface does not exceed the value set by the user. Use the counter to select or type the value of the maximum deviation in the Linear deviation field.</p>
 By linear deviation*	<p>The points are constructed in such a way that the angle between the tangent to the surface at the point and the edge of the approximating polyhedron connecting the point with the adjacent ones in U and V directions does not exceed the value set by the user. Use the counter to select or type the value of the maximum angular deviation in the Angular deviation field.</p>
 By grid around the defined point	<p>The points are positioned on the specified face's theoretical surface so that they form a grid around the defined point on the same theoretic surface. The defined point is the grid's center. The grid may be rectangular, concentric, or hexagonal. Examples of point groups built using this method, are shown in Fig. 2.5.15; the grid's center is marked by a circle. The grid's center position setting is described in Section 2.5.3.3.1 on p. 355, grid parameter setup is described in Section 2.5.3.3.2 on p. 357.</p>

* While choosing the methods by linear and angular deflection the density of points on the surface will depend on its curvature. The point density will be larger and smaller at segments with larger and smaller curvature, respectively.

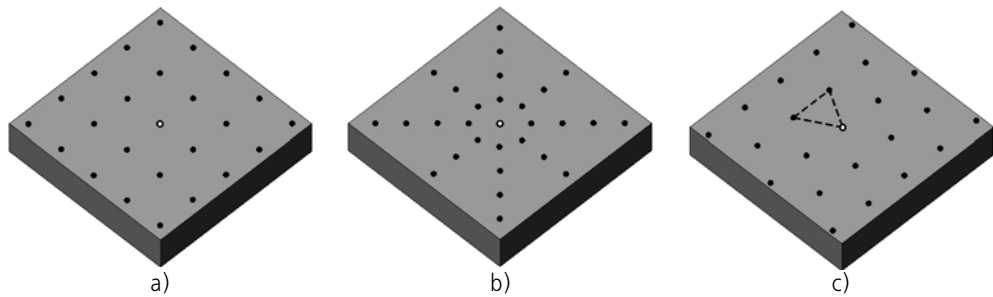


Fig. 2.5.15. Group of points constructed using the **By grid around the defined point**:
a) rectangular grid; b) concentric grid; c) hexagonal grid

4. Choose a point group construction variant using the **Allow for borders** option.
 - ▼ If the option is enabled, the point group will be constructed within the specified face's contour.
 - ▼ If the option is disabled, the group is constructed on the face's theoretical surface.
- Variants of point group construction on a face with a hole are shown in the figure.

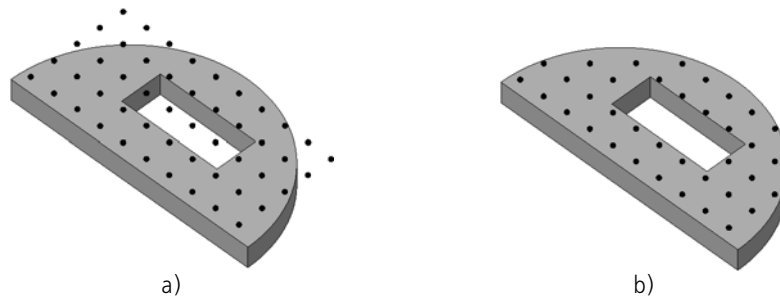


Fig. 2.5.16. An example of constructing point groups along a surface
a) group of points with the **Allow for borders option disabled**
b) group of points with the **Allow for borders option enabled**

2.5.3.3.1. Setting grid's center position during construction of a point group using the **By grid around the defined point** method

The grid's center position can be specified using two methods.

Method 1

Specifying parameters U and V of the point of the theoretical face's surface,

Specify the grid's center position by a mouse click on the face or enter the required values of the U and V parameters in percent in the **U Parameter, %** and **V Parameter, %** fields.

For more information on theoretical face surface, see Section 2.5.1.3 on p. 328.

You can also set the grid's center position by moving the "Center" hand of the point group's ghost (see Fig. 2.5.17).

Method 2

Specification of a point object on a chosen face within it. The following may be used as the point object:

- ▼ face vertex;
- ▼ a point in space provided that it is a **directly initial** or **directly derivative** with respect to the specified face¹.

Specify the required point object in the model window.



The **Object** button will be ticked, and the field next to the button will display the object name. This indicates the associative relation of the grid center with the selected object. Due to this relation, the grid center will follow the object in case it changes its position. If necessary, the relation can be disabled by clicking the **Object** button. The tick on the button will disappear, and the field will be cleared.

The grid center may be positioned both within the specified face's contour or outside it. If necessary, theoretical surface of a face will be extended (depends on the face's type). Examples of grid center positioning outside the specified face's contour with different statuses of the **Allow for borders** option can be found in the figure.

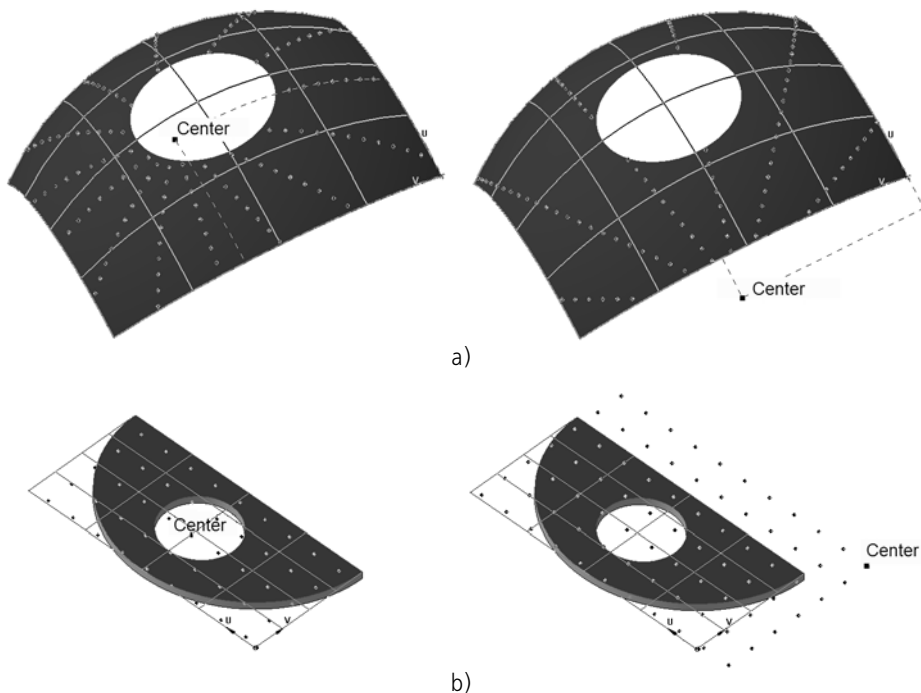


Fig. 2.5.17. Examples of grid center positioning outside the specified face's contour. when constructing point groups along a surface:
a) the **Allow for borders** option is enabled; b) the **Allow for borders** option is disabled

Note that if the grid center transcends the face's theoretic surface limits, it will limit the grid in its movement direction (Fig. 2.5.17, b).

1. In other words, the hierarchy of the relations between the point and the specified face must not contain any other objects (viewing object relations is covered in Section 2.1.2.7.2 on p. 114).

2.5.3.3.2. Setting up grid parameters during construction of a point group using the By grid around the defined point method

The grid's setup includes selection of the grid type and parameter settings.

Three types of grid are available: **rectangular**, **concentric**, and **hexagonal** (Fig. 2.5.15 on p. 355).

Choose the required grid type from the **Grid type** list (Fig. 2.5.18).

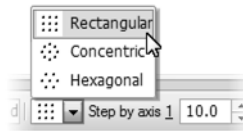


Fig. 2.5.18. The **Grid type** drop-down list.

The Property Bar will display controls for setting up the parameters of the grid of the selected type.

Availability of a certain grid type depends on the type of a face to which the group's points should belong.

- ▼ If a **spherical** face is used for construction of a point group, the following grid types are available: **Rectangular** and **Concentric**.
- ▼ If a **cylindric** face is used, the following grid types are available: **Rectangular** and **Hexagonal**.
- ▼ If a **conic**, **toroid**, or **custom** face is used, only the **Rectangular** grid is available.
- ▼ If a **flat** face is used, all grid types are available.

Detailed description of rectangular, concentric, and hexagonal grids is given below in this section.

Before completion of the command the system “remembers” the data entered for every type of grid, so you can switch between grid types without loss of their data.

If necessary, you can rotate the grid around the face's normal in the grid center by a custom angle. Such option is available, if a surface with a constant curvature is selected for construction of a point group (flat face or sphere).

The grid's rotation angle is calculated starting from the U direction counter-clockwise (see Section 2.5.1.3 on p. 328 for more information about the face's parametric representation).

Enter the value of the grid rotation angle in the **Rotation** field. It can either be positive – counter-clockwise rotation, or negative – clockwise rotation.

Rectangular grid

Control elements for setting rectangular grid parameters depends on the type of a face to which the group's points should belong.

- ▼ If a **flat**, **spherical** or **cylindric** face is used, the rectangular grid is characterized in steps along the first axis and along the second axis. The following fields appear on the Property bar: **Step along axis 1** and **Step along axis 2**.

- ▼ If a **conic**, **toroid**, or **custom** face is used, the rectangular grid is characterized by steps in U and V parameters. The following fields appear on the Property bar: **Step along U, %** and **Step along V, %**.

Enter the required grid parameter values in the fields or set them using the counter.

Concentric grid

The concentric grid is characterized by a number of radial rays and grid step in the radial direction. In order to set these parameters, fields in the Property bar are used: **Number of rays** and **Radial step**.

Enter the required grid parameter values in the fields or set them using the counter.

Hexagonal grid

The hexagonal grid is characterized by equal spacing between adjacent grid nodes.

Enter the required step values in the **Step** field or set it using the counter.

2.5.3.4. Point Group from File



To create a point group using the data from a file run the **Point Group from File** command.

The command creates a group of points with the coordinates read from a previously created data file (for more information about the data file, see Chapter VI).

The point parameters can be read from the following file types:

- ▼ Text file;
- ▼ Electronic spreadsheet in Microsoft Excel format;
- ▼ Electronic spreadsheet in Open Office format.



To define the file name click the **Source File** button. The standard Windows Open File dialog will appear on the screen. Select a file and click the **Open** button. The file name will appear in the field.

The values read from a file can be interpreted as rectangular, cylindrical or spherical coordinates.

The interpretation method is selected by means of the switches of the **Coordinate Type** group of the Properties panel.

The group points will be constructed in the coordinate system appropriate for the selected method of data interpretation. The location of that coordinate system relatively to the point coordinate system is detailed in Section 2.5.3.4.1.

2.5.3.4.1. Type of the point's coordinates

Point location can be set in rectangular (X, Y, Z), cylindrical (R, A, Z) or spherical (R, A, B) coordinates. The coordinate system corresponding to the type of coordinates is used to build the point.



Rectangular coordinate system coincides with the point coordinate system.



A cylindrical coordinate system is located relative to the coordinate system of a point in the following way:

- ▼ the cylindrical coordinate system origin coincides with the point coordinate system origin,
 - ▼ Z axis of the cylindrical coordinate system coincides with Z axis of the point coordinate system.
- The A coordinate value defines the point α rotation angle and is counted in the XY plane from the X axis counter clockwise (see Fig. 2.5.19).

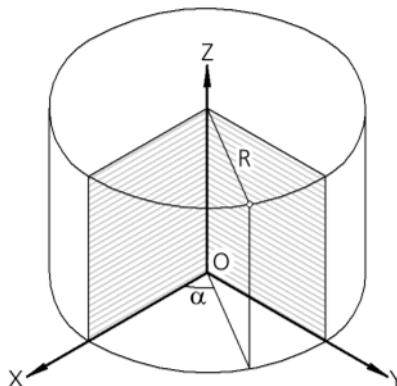


Fig. 2.5.19. Creation of a point by cylindrical coordinates; XYZ is the coordinate system of the point



The **spherical** coordinate system origin coincides with the point coordinate system origin.

The A coordinate value defines an azimuth angle α and is counted in the XY plane of the point coordinate system from the X axis counter clockwise (see Fig. 2.5.20). The OA ray is drawn in the XY plane at the azimuth angle to the X axis.

The B coordinate value defines the zenith angle β and is counted from the Z axis in the plane that extends through the Z axis and the OA ray (see Fig. 2.5.20).

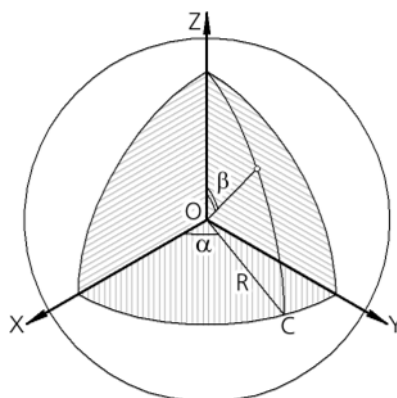


Fig. 2.5.20. Construction of a point by spherical coordinates; XYZ is the coordinate system of the point

2.5.3.4.2. Link of Point Group to Source File

The constructed point group remains linked to the source file.

Any modifications of the file will be shown by a red tick on the icon of the point group in the Design Tree.

To eliminate discrepancy between the constructed group and the file content run the **View—Reconstruct Model** command.

The command rearranges the group points according to the new parameter values.

If the source file is deleted or renamed, the point group icon will be marked by the error symbol in the form of an exclamation mark in a red circle in addition to the red tick.

If the point group is constructed with the **Scatter** option on, the built points will not be linked to the source file.

Applying the **Explode** command to the point group eliminates the link of the built points with the source file.

2.5.4. 3D lines

KOMPAS-3D supports creating the following 3D lines:

- ▼ segment,
- ▼ circle's arc,
- ▼ a cylindrical spiral,
- ▼ a conical spiral,
- ▼ polyline,
- ▼ a spline.

You can also import 3D curves saved in a file of the STEP AP203/STEP AP21 or IGES format in the model.

In addition, the program enables trimming curves and creating the following objects:

- ▼ a spline repeating the object form,
- ▼ curve by its two projections,
- ▼ curve that set in the parametric view,
- ▼ rounding of an angle formed by two curves,
- ▼ joining of two curves,
- ▼ Equidistant curve;
- ▼ contour,
- ▼ curves on the surface:
 - ▼ isoparametric curve,
 - ▼ group of isoparametric curves on the surface,
 - ▼ face outline line.
- ▼ spline on surface,
- ▼ projection curve,
- ▼ Intersectional curve of surfaces.

Commands for creating 3D lines and points are located in **Operations** menu, and the buttons to call them are on the **3D Lines** panel (Fig. 2.5.21).

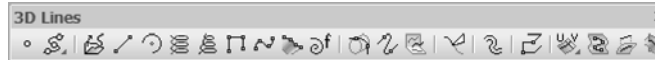


Fig. 2.5.21. **3D Curves** bar

The lines created can be used, e.g. as directing curves when creating swept features of corresponding form.



The model may create the so-called *curves from nowhere*. Such object emerge as a result of exploding arrays of certain data types (see Section 2.7.1.3.7 on p. 517).

2.5.4.1. Curve Display Parameters

Curve display parameters are vertex display style and line style. These parameters can be viewed and edited on the **Properties** tab of the Properties bar (Fig. 2.5.22).



Fig. 2.5.22. **Properties** Tab



The **Vertex Display** button controls the way 3D line vertices are displayed. If the button is pressed, list of **Vertex Styles** is available on the Property bar. You can select a vertex display style from this list.

Once the curve is constructed, its vertices will be displayed in the selected style.



For polylines with filleted vertices the end points of fillets are also marked (Fig. 2.5.23).

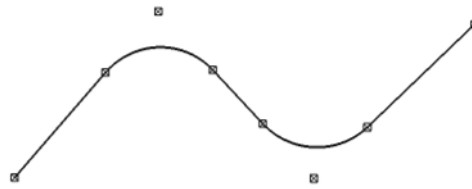


Fig. 2.5.23. Display of Polyline Fillet End Points



Vertices displaying button is available during creation of new curves in KOMPAS-3D, Ver. 9 and higher.

By default 3D line are visualized using a line with the *Basic* style applied. To change the current style use the **Line Style** list. The list contains the system styles of lines.



The parameters of current 3D line visualization set during its construction will be applied while creating further 3D curves until the end of the work session.

If necessary, line thickness can be changed by the user. To do this use command **Service — Parameters— System — Model Editor — System Lines**. A dialog will be displayed, in which you can set the required thickness settings.

2.5.4.2. Imported curve

To import a 3D curve (or 3D curves) into a KOMPAS-3D model file the following files may be used:

- ▼ file with the *igs*, *step* or *stp* extension – the curves from this file will be inserted into the current model,
- ▼ data file with *xls*, *ods*, *txt* or *csv* extension (for more information about the data file, see Appendix VI) – the curve will be built in the current model using the coordinates from the file.



The **Imported curve** command is used for the import operation.

After calling the command, a standard **Select file for opening** dialog appears, where you need to select the desired file.

The *igs*, *step* or *stp* file is analyzed for the presence of any 3D curves in it. If no 3D curves are identified in the file, the respective pop-up message will appear on the screen, while the file selection dialog will remain enabled.

After clicking the **Cancel** button in the dialog, the command finishes its operation without saving changes.



If there are curves in the source file, that form a chain, the prompt for unification of the curves will appear during the file reading. An affirmative answer implies that the curves will be unified wherever possible.

Once the import has been finished, icon of the imported curve or several icons of imported curves will appear in the Design Tree. The imported curves are “curves without history”.



Importing curves from files with *igs*, *step* and *stp* extensions has peculiarities described in Section 11.1.2.1.1 on p. 2006.

The coordinate system of the imported curve or curves will be the current coordinate system – absolute or local (see Section 2.8.3.3 on p. 597 for more details about selection of the current coordinate system).



If the coordinate system of an imported curve is not absolute but local, then you can change the position of the imported surface in the model by changing the position of the local coordinate system relative to the absolute coordinate system.

For detailed description of coordinate systems of objects, refer to Section 2.8.3.4 on p. 597.

2.5.4.3. Line segment

You can draw a section in space by the following methods:

- ▼ **By 2 Points,**
- ▼ **By point, length, and inclination angle method.**



In order to create a line segment, the **Line segment** command is used.

Upon calling the command, the construction controls appear on the Property bar (Fig. 48.4).

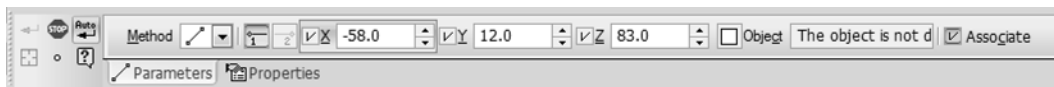


Fig. 2.5.24. Property bar when creating a line segment

Select an option of line segment construction from the **Method** list or using commands of the context menu.

Then specify line segment parameters and objects used for construction.

After all required data are specified, a ghost of the line segment appears on the screen. Until construction is finished, you can change the line segment parameters by activating the required control element on the Property Bar.



To change objects used for the line segment construction, click **Reselect** button on the Special Control Panel and re-select objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the line segment, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361). This tab also shows the name of the line segment coordinate system name (object coordinate system is detailed in Section 2.8.3.4.1 on p. 598).



To finish the construction click the **Create object** button on the Advanced Control Panel.

All methods of line segment creation provide for automatic creation activated by default. For more information on automatic and manual creation of objects refer to Section 1.4.2.10 on p. 93.

2.5.4.3.1. Construction using 2 points



The **By 2 points** method makes it possible to build a line segment by specifying the starting and the ending vertex successively.

To create a line segment, complete the following steps.



1. Set the vertex of the segment. The **Starting vertex** switch will be enabled by default. There are several methods to set up a vertex.

- ▼ Setting the vertex coordinates in space or linking it to an existing point object. Specifying of point coordinates and linking the point to a point object are similar to the procedures used during point creation by the **By Coordinates** method (see Section 2.5.2.2 on p. 336).



- ▼ Creating a segment vertex using the methods of point construction provided by the **Point** command. For this purpose, click the **Point construction** button on the Special Control Panel. The detailed information on point construction is given in Section 2.5.2.1 on p. 334.



After setting the segment's starting vertex, the **Ending vertex** switch will become enabled automatically.

2. Set the ending vertex of the segment. The segment's ending vertex is specified in save way as the starting vertex.

2.5.4.3.2. Construction by point, length, and inclination angle



By point, length, and inclination angle is the method of the segment construction with specification of the segment's plane and setting the following parameters: starting vertex, length, and inclination angle of the segment.

The plane of the segment is parallel to a **base plane**. The following can be used as a base plane:

- ▼ Coordinate or auxiliary plane
- ▼ Planar face;
- ▼ Sketch plane

To create a line segment, complete the following steps.



1. Set the vertex of the segment. The **Starting vertex** switch will be enabled by default. There are several methods to set up a vertex.

- ▼ Setting the vertex coordinates in space or linking it to an existing point object. Specifying of point coordinates and linking the point to a point object are similar to the procedures used during point creation by the **By Coordinates** method (see Section 2.5.2.2 on p. 336).



- ▼ Creating a segment vertex using the methods of point construction provided by the **Point** command. The **Point construction** button is located on the Special Control Panel. The detailed information on point construction is given in Section 2.5.2.1 on p. 334.

By default, the base plane is the one of the coordinate planes. The base plane is selected automatically by the system depending on the model position with respect to the screen plane. The name of the selected coordinate plane is shown in the **Base Plane** field.

After selecting the starting vertex, the segment plane position is automatically defined: by specified point and base plane. The segment's ghost will be displayed.



If necessary, you can select another base plane. To do this, enable the **Base Plane** switch on the Property bar or call the command from the context menu, and select the required object in the Design Tree or the model window (a sketch must be selected in the Design Tree).



You may start the segment construction by defining a base plane: activate the **Base plane** switch, select the required object, then define the starting vertex.



After setting the segment's starting vertex, the **Ending vertex** switch will become enabled automatically.

2. Set the ending vertex of the segment. There are several methods to set up the ending vertex:
 - ▼ setting by the mouse in the model window or linking it to a point object (see Section 2.5.2.2 on p. 336);
 - ▼ setting the segment's length or inclination level in the base plane.
Set the segment's length.

The segment's length can be entered from the keyboard. You can also specify the point's position using the mouse in the model window. In this case, the segment's inclination angle is defined automatically.

Set value of the segment's inclination angle.

The angle of the segment's inclination is counted off from the abscissa axis of coordinate system of the base plane. The angle's value can be entered from the keyboard. You can also specify the point's position using the mouse in the model window.



If the ending vertex has an associative link to a point object, the **Length and Angle** fields will not be available for editing.

2.5.4.4. Circular Arc

You can construct a circular arc in space by the following methods:

- ▼ **By 3 Points,**
- ▼ **By Center and Radius,**
- ▼ **By 2 Points with Direction,**
- ▼ **Tangent to Curve.**



To create a circular arc in space, call the **Circular Arc** command.

Select an option of point construction from the **Method** list or using commands of the context menu. The elements for managing construction will appear on the Properties Bar (Fig. 2.5.25).

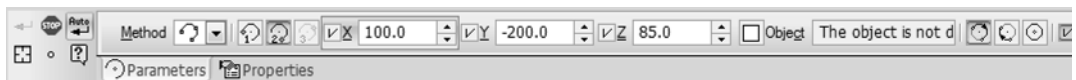


Fig. 2.5.25. Property Bar in case of circular arc creation by 3 points

Then specify arc parameters and objects used for construction.

After specifying one of the arc parameters, you will automatically transfer to the next parameter.

Points are the construction objects in all methods. The following ways may be used to define an arc point:

- ▼ specify coordinates in space or select the point's position by mouse in the model window;
- ▼ link to an existing point object;
- ▼ create a point using the point creation methods of the **Point** command without leaving the arc creation process (see Section 2.5.2.11 on p. 346).

Selecting of the point object when the **Associate** option is on will result in creation of associative relation of the point with this object.



The presence of relation is manifested by displaying a checkmark on the **Object** button and the name of the point object in the **Object** field on the Property Bar. In case of any change of the object position, the arc point will follow the object.



To cancel the associative relation with the point object, click the **Object** button. The tick on the button will disappear.



To restore an associative relation of a point and a point object, enable the **Auto association** option and specify the required object anew.

Parameter input and object specification for each method is detailed in Sections 2.5.4.4.1–2.5.4.4.4.



Point coordinates are specified in the arc coordinate system (for object coordinate system, refer to Section 2.8.3.4 on p. 597).

After all required data are specified, a ghost of circular arc appears on the screen. Until construction is finished, you can change the arc parameters by activating the required control element on the Property Bar.



If required you may construct a circle. To do this, enable the **Circle** switch in the **Arc** group. It is possible for all methods of arc construction.

Before completion of the command, the system “remembers” the parameters and objects specified in each construction method, therefore you can switch between the methods without loss of these data.



To change the objects used for arc construction by the selected method, click the **Select Other** button on the Special Control Panel and repeat selection of objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the arc, set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361). This tab also shows the name of the arc coordinate system name (object coordinate system is detailed in Section 2.8.3.4.1 on p. 598).



To finalize the construction, click the **Create Object** button on the Special Control Panel.

All methods of circular arc creation provide for automatic creation activated by default. For more information on automatic and manual creation of objects refer to Section 1.4.2.10 on p. 93.

2.5.4.4.1. Creation by 3 Points



By 3 points is a method of circular arc creation by three points.

To create a circular arc by this method, specify arc start point **1**, then arc end point **2** and intermediate point **3** on the arc. After selecting point **1**, a circle ghost will appear on the screen, after selecting point **2**, an arc ghost will appear. After specifying intermediate point **3**, the arc is defined completely.

- ▼ You can select an arc point by setting its coordinates in space or linking it to an existing point object. Specifying of point coordinates and linking the point to a point object are similar to the procedures used during point creation by the **By Coordinates** method (see Section 2.5.2.2 on p. 336).
- ▼ You can specify an arc point by creating it. In this case, use the point creation methods of the **Point** command. See Section 2.5.2.1 on p. 334 for details.



By default, a circular arc passes from point **1** to point **2** through point **3**. In this case, the **Default Arc** switch is active. To create a supplementary arc (that does not pass through point **3**) or a circle, enable the required switch.

2.5.4.4.2. Creation by Center and Radius



By Center and Radius is a method of circular arc creation by specifying its plane and parameters: center of the arc, starting and ending angles, radius.

The plane of a circular arc is parallel to a **base plane**. The following can be used as a base plane:

- ▼ Coordinate or auxiliary plane
- ▼ Planar face;
- ▼ Sketch plane

To construct an arc, do the following.

1. Select centerpoint **1** of the arc.
- ▼ You can select an arc point by setting its coordinates in space or linking it to an existing point object. Specifying of point coordinates and linking the point to a point object are similar to the procedures used during point creation by the **By Coordinates** method (see Section 2.5.2.2 on p. 336).
 - ▼ You can specify an arc point by creating it. In this case, use the point creation methods of the **Point** command. See Section 2.5.2.1 on p. 334 for details.

By default, the base plane is the one of the coordinate planes. The base plane is selected automatically by the system depending on the model position with respect to the screen plane. The name of the selected coordinate plane is shown in the **Base Plane** field.



If necessary, you can select another base plane. To do this, enable the **Base Plane** switch and select the required object in the Design Tree or the model window (a sketch must be selected in the Design Tree).



You may start the arc construction by defining a base plane: activate the **Base plane** switch, select the required object, then define point **1**.

After selecting point **1**, the arc plane position is automatically defined: by specified centerpoint and base plane. A ghost of an arc circle with centerpoint **1** will appear on the screen.



2. Enter the arc radius/diameter. By default the system waits for the radius to be entered. In this case, the **Radius** switch is active in the **Parameter** group. To enter the diameter, activate the **Diameter** switch. A ghost of a circle with the specified dimension will appear on the screen.



3. Select the arc start point **2** by specifying a start angle of the arc. The start angle is counted from the abscissa axis of the base plane's coordinate system. You can enter an angle value from keyboard or link the arc start point to an existing point object. A ghost of an arc with the specified start angle will appear on the screen.

4. Select the arc point **3** by specifying the end angle. The end angle is specified in the same way as the start angle.



After performing the above steps, the arc is defined completely: arc plane position, center point, start and end angles, radius are specified.



By default, a circular arc is created clockwise from point **2** to point **3**. In this case, the **Default Arc** switch is active. To create a supplementary arc or a circle, enable the required switch.



In case of creation of a complete circular arc, after selecting point **1**, the system immediately proceeds to specifying radius (point **4**).

2.5.4.4.3. Creation by 2 Points with Direction



By 2 Points with Direction is a method of circular arc creation by two points with specifying a direction of a tangent vector in one of them. The direction of a tangent vector is selected by means of a vector or guide objects. The following objects can be used as guide objects.

▼ Straight object:

- ▼ line segment of a sketch,
- ▼ polyline segment,
- ▼ coordinate or auxiliary axis,
- ▼ straight line edge.

Direction, defined by a straight object — a line parallel to the object.

▼ Planar object:

- ▼ coordinate or auxiliary plane,
- ▼ planar face,
- ▼ Sketch plane

Direction, defined by a plane object — a line perpendicular to the object.

▼ Curve (if the start point or the end point of the arc belongs to this curve).

The direction defined by the curve is tangent vector in the point of tangency. Thus, you may construct an arc mated with an arbitrary curve: 3D spline or polyline, edge etc.

To construct an arc, do the following.

1. Specify start point **1** of the arc.
 - ▼ You can select an arc point by setting its coordinates in space or linking it to an existing point object. Specifying of point coordinates and linking the point to a point object are similar to the procedures used during point creation by the **By Coordinates** method (see Section 2.5.2.2 on p. 336).
 - ▼ You can specify an arc point by creating it. In this case, use the point creation methods of the **Point** command. See Section 2.5.2.1 on p. 334 for details.



2. Select end point **2** of the arc. Specify point **2** in a similar way as point **1**. After specifying point **2**, the system will proceed to the selection of direction for this point: the **Direction in Point** switch corresponding to point **2** will be enabled.

3. Specify the direction. To do this, select the appropriate guide object in the Design Tree or in the model window (a sketch must be selected in the Design Tree) or build a vector (see Section 2.8.5 on p. 613).



To build a vector click the **Construction of Vector**. This button is located on the Advanced Control panel.

The name of the selected object will be displayed in the **Direction in Point** field. If the direction is set by means of a vector, the field will contain the word "Vector".

A ghost of the arc with points **1**, **2** and a tangent vector in point **2** showing the specified direction will appear on the screen.



For the arc mated with a curve, you must specify the point on this curve: the start point, the end point or the point created by **On Curve** or **On Intersection** methods (see Section 2.5.2.1 on p. 334), and the curve itself as a guide object in this point.



The direction may be specified only in one point of the arc, i.e. start point or end point. That is why if the direction in one of this points has already been specified, the selection of the object for the first point should be canceled after specifying the direction in the other point.

After specifying points **1**, **2** and selecting the direction, the arc is defined completely: the system creates the arc plane and calculates the position of the center point and the radius.

The arc plane is created by a straight object and a point. A tangent vector is used as a straight object.

The center point C of the arc lies on the intersection of the axis c and perpendicular bisector m to the segment connecting points **1** and **2** (Fig.2.5.26). Axis c is perpendicular to the vector of direction in the point.

Radius is defined as the distance between the centerpoint C and the point **1**.

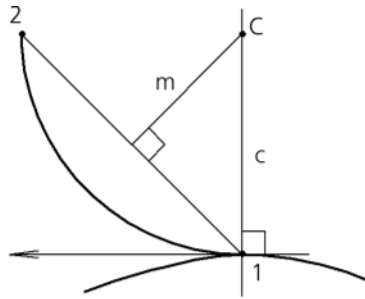


Fig. 2.5.26. Specifying centerpoint and radius



By default, a circular arc runs from point **1** to point **2** in the direction similar to the direction of a tangent vector. In this case, the **Default Arc** switch is active. To create a supplementary arc or a circle, enable the required switch.

2.5.4.4.4. Creation of Arc Tangent to Curve



Tangent to Curve is a method of circular arc creation by specifying a curve the arc circle should be tangent to, and the arc parameters: the center, starting and ending angles.

The following may be used as a curve:

- ▼ coordinate or auxiliary axis,
- ▼ a 3D curve,
- ▼ sketch lines,
- ▼ edge.

To construct an arc, do the following.

1. Select the curve the arc circle should be tangent to. To do this, select the desired object in the Design Tree or in the model window. The selected object's name will be displayed in the **Curve** field. A ghost of a circle tangent to the specified curve will appear on the screen.
2. Select centerpoint **1** of the arc.
 - ▼ You can select an arc point by setting its coordinates in space or linking it to an existing point object. Specifying of point coordinates and linking the point to a point object are similar to the procedures used during point creation by the **By Coordinates** method (see Section 2.5.2.2 on p. 336).
 - ▼ You can specify an arc point by creating it. In this case, use the point creation methods of the **Point** command. See Section 2.5.2.1 on p. 334 for details.

After specifying point **1**, specify the location of arc plane and radius (point **4**).

The plane is created by a point and a straight object. A straight object is a tangent line passing through the point of a specified curve that is closest to the centerpoint of the arc.

The calculated value of radius is displayed in the **Radius** field. If required, you can enter a desired radius value into this field. The system will calculate the new position of centerpoint **1**.



If the point **1** is associatively related to a point object, then the value in the **Radius** field cannot be changed.



You can use a circle diameter to create an arc. To do this, enable the **Diameter** switch in the **Parameter** group.

3. Select the arc start point **2** by specifying a start angle of the arc. The start angle is counted from the ray directed from the center of the arc to the tangency point. You can enter an angle value from keyboard or link the arc start point to an existing point object. A ghost of an arc with specified start angle and point **4** will appear on the screen.
4. Select the arc point **3** by specifying the end angle. The end angle is specified in the same way as the start angle. A ghost of the arc with specified points **1, 2, 3, 4** appears on the screen.



If points **2** and **3** are associatively related to point objects, then the fields for entering angle values are not accessible, they show angle values calculated by the system.



After performing the above steps, the arc is defined completely: arc plane position, center point, start and end angles, radius are specified.



By default, a circular arc is created clockwise from point **2** to point **3**. In this case, the **Default Arc** switch is active. To create a supplementary arc or a circle, enable the required switch.



In case of creation of a complete arc, after specifying point **1**, a circular arc is defined completely: the **Create Object** button on the Special Control Panel becomes accessible.

2.5.4.5. Spirals. General construction methods

Spiral construction commands are accessible if some planar object is selected in a model window: an auxiliary or projection plane, flat face of a part. This item is a reference object for a spiral.

After any of commands to construct a spiral is applied you must specify its parameters in the fields on the Property bar.

All the parameters are displayed on the screen as a ghost of a spiral.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the spiral, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361).



To fix this spiral in the model, click the **Create Object** button on the Special Control Panel.



The created spiral will appear in the model window, and the spiral's icon will appear in the Design Tree.

2.5.4.6. Cylindrical Spiral



To create a cylindrical spiral, apply the **Cylindrical Spiral** command.

2.5.4.6.1. Construction Method

Select the method for spiral creation: **By Pitch and Number of Revolutions**, **By number of Revolutions and Height** or **By Pitch and Height** selecting it from the **Construction Method** list.

2.5.4.6.2. Number of revolutions, pitch, height

Enter the number of revolutions of the spiral in the **Number of Revolutions** field. Number of revolutions can be non-integral. If the spiral construction method **By Pitch and Height** is selected, then **Number of Revolutions** field becomes inaccessible for value input and it displays the calculated number of revolutions.

Enter a pitch of spiral in the **Pitch** field. If the spiral construction method **By Number of Revolutions and Height** is selected, then the **Pitch** field is inaccessible for data input and it displays the calculated value of the pitch.

If the spiral construction method **By Pitch and Height** or **By Number of Revolutions and Height** is selected, the group of options **Height** for setting the spiral height becomes accessible.

By Value method

The height of spiral equals the value specified in the **Dimension** field.



Activate the **By Value** switch in the **Height** group and enter the spiral height into the **Dimension** field.

By Object method

The height of the spiral is automatically determined by location of the object specified by a user (an edge or a point in the model window).



Activate the **By Object** switch in the **Height** group and specify vertex, face, plane or surface in the model window. The name of the object will be displayed in the **Object** reference field.

Enter the required distance between the element's end plane and the object in the **Dimension** field

In order to draw a spiral exactly to the vertex or the surface, enter a zero value.

If the distance to an object exceeds the zero value, it can be applied both in the direction of spiral construction (in this case, the spiral will be extended "off" the object for the distance specified), and against the direction of spiral construction (in this case, the spiral won't reach the object for the distance specified). To specify direction of measuring the distance to the vertex, activate the **Before Object** or **Behind...** switch in the **Type** group.



By Flat Curve method

Spiral's height is defined automatically by the length of the indicated sketch line (spiral has the same form as the line while the spiral axis is a cop of the line resulting from the parallel transfer of the sketch line where its initial point matches the spiral's anchor point. The plane of used sketch must be perpendicular to the base face of the spiral.



Activate the **By Flat Curve** switch in the **Height** group and select the required sketch line in the model window.



The object name will be displayed in the reference field beside the **Object** switch.

If sketch plane and spiral axis plane are not parallel, spiral will automatically rotate so that these planes become parallel.

2.5.4.6.3. Direction



By default, a spiral will be constructed outwards in relation to the base face. This is **Straight Direction**. To direct the spiral to the other direction, activate the **Reverse Direction** switch in the **Direction to Construct** group.



If the spiral height is defined by object or sketch line, then changing of the spiral construction direction is impossible (**the Direction to Construct** group is inaccessible).

2.5.4.6.4. Direction of Coiling



Select the direction of the spiral's coiling — **Right** or **Left**.

To do this, activate the relevant switch in the **Direction of Coiling** group.



2.5.4.6.5. Starting Angle

Enter the start angle of the spiral in the **Angle** field. It is the spiral rotation angle around the perpendicular to the reference surface.



The angle is measured in relation to the abscise axis of the base face's coordinate system.

2.5.4.6.6. Spiral diameter

Options to control the spiral's diameter are located on the Properties Bar's **Diameter** tab.

Select a method to define the spiral diameter. For this, you should activate the corresponding switch in the **Diameter** group (see Table). 2.5.5).

Tab. 2.5.5. Methods to define the diameter of a cylindrical spiral

	Value of Option	Procedure of defining the spiral's diameter. Work sequence
	By Value	Diameter of a spiral is equal to the value specified by a user. To select this variant enter the spiral's diameter into the Diameter 1 field.
	By Object	The diameter of the spiral is automatically defined by size or by location of the object specified by a user (see below). To select this variant, specify in the model window the object to define the spiral's diameter. The object name will be displayed in the Object 1 reference field, and the Diameter 1 field will read the calculated value of the diameter.

Object to Define Spiral Diameter

- ▼ A point can be selected as the object (a part vertex, a coordinate system origin, a line end or a point in a sketch, etc.). In this case, the spiral diameter is assumed to be equal to the distance from the spiral center to the projection of the specified point on the spiral's base plane.
- ▼ You can select a part edge or sketched image having the shape of a circle or arc, as the object. In this case, the spiral diameter is assumed to be equal to the diameter of the selected arc or circle.

2.5.4.6.7. Spiral Location

The cylindrical spiral ghost with the specified parameters will appear in the model window. Spiral Anchor Point by default is located at the coordinate system origin of a face or plane, on which the spiral is created.

To place the spiral in the desired position on a face or plane, unfix the **t** field on the **Property Bar**. Select the spiral position with the mouse or enter the spiral center coordinates in the **t** field.

Once the spiral's construction in the model has been finished, aside from the spiral itself, a sketch containing one object (the spiral's anchor point) is also created. That sketch is located on the plane or flat face selected for a spiral construction. If required, you can control the position of an already created spiral by changing the position of the point in the sketch (see Section 2.14.3.5.1 on p. 791).

2.5.4.7. Conical Spiral



To create a conical spiral, enable the **Conical Spiral** command.

Construction method, number of revolutions, pitch, height, direction of coiling and construction of a conical spiral are to be set in the same way as for a cylindrical spiral (see Sections 2.5.4.6.1–2.5.4.6.5).

A conical spiral differs from a cylindrical one by presumption that its initial diameter won't be equal to the final diameter and the spiral height definition using a flat curve is not accessible.

Options to manage the initial and final diameters are located on Properties Bar's **Diameter** tab.




2.5.4.7.1. Initial and Final Diameters

Select a method to set the spiral's initial and final diameters. For this, activate the corresponding switch in the **Start Diameter** and **Final Diameter** groups (see Table 2.5.6).

Tab. 2.5.6. Methods to set the initial diameter for a conical spiral



Option value	Procedure of defining the diameter.
Starting Diameter (Ending Diameter)	Work sequence
By Value	Diameter is equal to the value specified by user. To select this variant, enter the initial (final) diameter of the spiral in the Diameter 1 (Diameter 2) field.

Tab. 2.5.6. Methods to set the initial diameter for a conical spiral

	Option value	Procedure of defining the diameter. Work sequence
 	By Object The diameter is automatically defined by the size or position of the object specified by a user (for the object defining the spiral's diameter — see p. 473). To select this variant, specify the object required in the model window. The object name will be displayed in the Object 1 (Object2) reference field, and the Diameter 1 (Diameter 2) field will read the calculated value of the diameter.	
	By Slope of Generating Line* The final diameter is calculated by the specified values of the initial diameter and the slope of the conical spiral's generating line. To select this variant, enter in the Angle field the value of the generating line's slope to the spiral axis. Select direction of the generating line's slope by enabling the corresponding switch in the Slope of Generating Line group (see Table 2.5.7). The calculated value of the final diameter will appear in the Diameter 2 field.	

* For final diameter only

Tab. 2.5.7. Slope direction of conical spiral generatrix

	Value of option	Construction result
	Outside	The spiral diameter is increasing along the construction direction.
	Inside	The spiral diameter is decreasing along the construction direction.

2.5.4.7.2. Spiral Location

The conic spiral ghost with the specified parameters will appear in the model window. Spiral Anchor Point by default is located at the coordinate system origin of a face or plane, on which the spiral is created.

To place the spiral in the desired position on a face or plane, unfix the **t** field on the **Property Bar**. Select the spiral position with the mouse or enter the spiral center coordinates in the **t** field.

2.5.4.8. 3D Splines and Polylines General construction methods

3D polyline and spline are pointwise set curves. The points defining these curves are referred to as **vertices**.

A 3D polyline consists of lines that consequently connect the specified vertices. You can round vertices of a polyline by circle arcs.

A spatial spline is defined by the set of vertices and its order. The spline shape depends on the position of the defined vertices in space. Depending on the type a spline may go directly through the specified vertices or at a distance from them. The distance to each vertex depends on its weight.

For details on curve order and point weight refer to Section 2.5.1.4 on p. 329.

After running the curve construction command set the parameters of its vertices.

A polyline vertex can be constructed as follows:

- ▼ **By points** (see section 2.5.4.8.3 on p. 379);
- ▼ **By Axes** (see Section 2.5.4.8.4 on p. 381);
- ▼ **By Objects** (see Section 2.5.4.8.5 on p. 383).

You can do any of the following to define a vertex coordinates:

- ▼ entering coordinate values into the parameter table of polyline vertices (see Section 2.5.4.8.1 on p. 376),
- ▼ creating a special point (see Section 2.5.2.11 on p. 346),
- ▼ linking a vertex to a reference point (see Section 2.5.4.8.6 on p. 384).

While each vertex is constructed, the line connecting it to the previous one – the **vertex construction line** – is being displayed.

2.5.4.8.1. Vertex Parameter Table

Every vertex of a polyline or a spline is characterized by a set of parameters. Their values are displayed in the Table of Vertices (Fig. 2.5.27).

This table is called **Vertices Coordinates** and is located on the Properties Bar's **Parameters** tab.

Every vertex is specified by a table row.

The following parameters are common for all the spline and polyline:

- ▼ Vertex number,
- ▼ 3D coordinates of the vertex;
- ▼ Relation mark is the vertex is associated with a point object;
- ▼ Relation and vertex marks if the vertex was created using the **Point** command (see Section 2.5.2.11 on p. 346) by any method, except for the **By coordinates (XYZ)** method.

The specific parameters are the following:

- ▼ for a polyline — vertex rounding radius (see Section 2.5.4.10.2 on p. 398);
- ▼ for a spline (when creating a spline by poles; see Section 2.5.4.9.1 on p. 387):
 - ▼ weight;



- ▼ relation icon and system pole icon (for more about system pole see Section 2.5.4.9.3 on p. 389).

N	<input type="checkbox"/>	X	Y	Z	Weight
1	<input checked="" type="checkbox"/>	88.0	-73.0	-16.0	1.0
2	<input checked="" type="checkbox"/>	55.0	-58.0	4.0	1.0
3	<input checked="" type="checkbox"/>	0.0	0.0	0.0	1.0
4	<input checked="" type="checkbox"/>	-63.0	77.0	-14.0	1.0
5	<input checked="" type="checkbox"/>	-45.0	35.0	10.0	1.0
6	<input checked="" type="checkbox"/>	-74.0	57.0	17.0	1.0
7	<input type="checkbox"/>	-79.0	83.0	-4.0	1.0

a)

N	<input type="checkbox"/>	X	Y	Z	Radius
2	<input checked="" type="checkbox"/>	-6.0	-40.0	-63.0	0.0
3	<input type="checkbox"/>	100.0	-40.0	-63.0	10.0
4	<input checked="" type="checkbox"/>	100.0	-40.0	-41.0	10.0
5	<input type="checkbox"/>	100.0	-8.0	-41.0	10.0
6	<input checked="" type="checkbox"/>	80.0	20.0	22.0	10.0
7	<input checked="" type="checkbox"/>	100.0	24.0	26.0	0.0

b)

Fig. 2.5.27. Table of Vertices' Parameters a) spline, b) polyline

The table rows are formed automatically during vertex construction in the model window. At the same time, creating or editing the table row shall involve creation of a new table or will change the parameters for the current curve vertex.



Curve vertex coordinates are specified in curve's coordinate system (for object coordinate system, refer to Section 2.8.3.4.1 on p. 598).

A detailed description on methods of operation with the table of vertex parameters is provided in Section 2.5.4.8.2 on p. 377.

2.5.4.8.2. Methods of operation with the table of parameters

Vertex parameter table controls and table context menu commands allow the following manipulations:

- ▼ replacing the entries in all cells of the selected column with the zero value or value from the first cell;
- ▼ fill in the table with a set of values from an external text file;
- ▼ save the contents of the parameters table into a text file for a future reuse;
- ▼ create new vertices;
- ▼ modify parameter values of existing vertices in the table;
- ▼ delete the curve's vertices;
- ▼ Allows deleting the connection of a vertex to a point object;
- ▼ Cancel spline mating with the model object.

The entries in all cells of the selected column replacing

To select a column, click its header by the mouse. You can also pick a cell by the mouse, which belongs to the column to be selected, and call the context menu **Select Column** command.

The **Set Value to Zero** command available in the table's context menu allows replacing the entries of all cells of the selected column with the value *0.0*.

The **Copy Value by 1st Cell** command available in the table's context menu allows replacing the entries of all cells of the selected column with the value entered in the first cell.



The **Copy Value by 1st Cell** command is available if the curve does not contain vertices related with point objects.

Reading the vertices file

While constructing a spline or a polyline you can set the vertex parameters using the data previously saved in a file. For more information about the data file, see Chapter VI.



To construct a curve using data from a file saved on the disk click the **Read from File** button. Select the required file in the Open A File standard dialog on the screen. Table of parameters shall be filled in with the data from this file. The ghost of the polyline constructed by the table data will appear in the Model window.



The **Read from File** button is accessible only when a spline parameters table is empty.

Saving Vertex Parameters to File



To save the vertices parameters into a file you should click the **Save to File** button.

The standard Windows save file dialog will appear on the screen. In this dialog you need select a file name and the directory to save. By default the file shall be assigned the *txt* extension.

Coordinates can be saved into text file only if at least one row is filled in the parameter table.

Creating a new vertex in the parameters table



To create a vertex before an existing one, select the line with this vertex ID and click the **Create New Vertex** button or the *<Insert>* key.

In the model window a new vertex shall appear. Its ID number will be equal to the number of the current vertex. Numbers of vertices starting from the one before which the new one was inserted will be increased by one. Parameter values of the created vertex will be the same as those of the vertex before which it is inserted. The newly created a vertex parameters line will be active.

Changing of the vertex parameters in the table of parameters

To change the curve vertex parameter, activate the desired cell of parameters table by double clicking the mouse and enter the new value to it. To move between cells, use arrow keys.



Coordinates can be entered only for the vertices of a curve constructed using the **By Point** method and not related to a point object.

Deleting Vertices from the Table



To delete a vertex, select the row to be deleted and click the **Delete Vertex** button or press the <Delete> key.

Using the <Shift>+ <arrow> keys, you can select several rows and delete them at once.

Deleting a relation between a vertex and a point object



To delete the relation of a vertex with the point or point object constructed for it highlight the necessary row in the table and click on the relation mark for the vertex in the **Relation** column or run the **Delete Relation** command from the table context menu. The relation will be deleted, the vertex coordinates won't be changed.



Be careful when deleting a relation, since this action cannot be undone. To restore the relation you should edit the vertex by recreating or specifying the required point.

Canceling spline mating with the model object



If the spline vertex has the attribute of a system pole, the spline mating can be deleted by canceling the attribute (mating splines is detailed in Section 2.5.4.9.2 on p. 388). To cancel spline mating highlight the system pole row underlying the mating in the table, click on the system pole mark in the **Relation** column or run the **Delete Relation** command from the table context menu. The mating will be cancelled whereas the system pole attribute will be dissociated from the vertex. The vertex coordinates will remain unchanged.



Be careful when deleting a mating, since this action cannot be undone. You will have to redo the mating in order to restore it.

2.5.4.8.3. Construction by Point

To construct a spline or polyline vertex by points select option **By Points** from the **Construction Method** drop-down menu.

The following methods can be applied to define vertex coordinates:

- ▼ Picking a vertex position by the cursor in the Model window,
- ▼ Linking the vertex to a point object;
- ▼ Construction of a special point;
- ▼ Inputting data in Table of Vertex Parameters cells,
- ▼ Reading parameter values from the vertex parameter file;

Selection of the vertex by cursor and linking the vertex with the point object are covered further in this section. The use of vertices table and reading the contents of the vertex parameter from a file are described in Section 2.5.4.8.1 on p. 376 whereas point construction is covered in Section 2.5.2.11 on p. 346.

Specifying vertex position in the Model window

The position of a polyline or spline vertex can be defined by cursor in the Model window. In this case, the vertex coordinates will be defined by:

- ▼ The cursor position on the screen plane;
- ▼ The screen plane orientation in the model space;
- ▼ Constraints (fixes), which might restrict coordinate modifications.

To fix a coordinate value, use the switch located beside this coordinate input field. By default the coordinate is non-fixed. For this purpose, the switch has a checkmark. To fix a coordinate value, you can do the following:

- ▼ Click the switch;
- ▼ Press <Enter> when the cursor is in the coordinate input field.

After fixing the coordinate value, the mark on the switch will be a “cross”.

To finish building the polyline vertex click in the Model window or fix all of the three vertex coordinates.

A new vertex will appear in the Model window. Its parameters line will be added to the Vertices Parameters Table.

After that, the system will be waiting for another vertex to be selected.



The state of a coordinate value being fixed is maintained only when inputting the current vertex coordinates. After creating it, the coordinate will be unfixed.

While constructing a polyline you can create several successive points with the same value of one of the coordinates (for example, when creating vertices in the same plane). For this, do the following.

1. Define and fix the value of the concerned coordinate.
2. Click the **Remember Status** button on the Special Control Panel.
3. Keep selecting the positions of polyline vertices. The fixed status of the coordinate will be preserved.
4. When finished with creating vertices with the same coordinate value, release the **Remember Status** button.



Linking the vertex to a point object

To link the vertex to a point object highlight the necessary point object with the cursor.

To specify a point object, hover the cursor over the object. When the symbolic notation of a line appear next to the cursor, click the left mouse button.

The new vertex will appear in the Model window. The row with its parameters in the Vertex Parameters Table will be automatically filled with the coordinates of the selected point.

If the **Associate** option is on, the vertex is associated with the selected point object (see Section 2.5.4.8.6 on p. 384).

If the curve vertex was created with the **Associate** option off, the curve vertex will coincide with the selected point object, but will not be related to it - i.e. while moving the point object the vertex position will remain unchanged.

2.5.4.8.4. Construction by Axes

Construction of a spline or a polyline by axes is limited by the following restriction to the direction of the vertex line construction: the line is constructed parallel to the selected axis of the polyline coordinate system (object coordinate system is covered in Section 2.8.3.4.1 on p. 598).

To select a coordinate axis, which the polyline segment will be parallel to, use any of the following methods:

- ▼ Select the required variant from the **Construction Method** drop-down list,
- ▼ Call the respective context menu command,
- ▼ Use the coordinate axes displayed next to the created a vertex in the Model window (see Fig. 2.5.28).

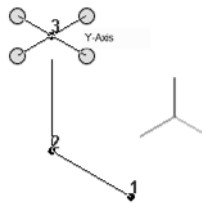


Fig. 2.5.28. Specify a coordinate axis in the Model window

For the last instance, the variant, corresponding to the specified direction, is automatically selected from the **Construction Method** drop-down menu. Once the coordinate axis is selected, if you move the screen cursor, the line of vertex construction, parallel to that axis, will be displayed.

The following methods can be used to define the vertex coordinates on the construction line:

- ▼ setting offset of the vertex to be created against the previous surface or a preset surface.
- ▼ Specifying vertex position in the Model window;
- ▼ Linking the vertex to a point object.

After that, the system will be waiting for another vertex to be selected. The direction of the vertex construction line will be saved. To modify it, use the following methods:

- ▼ Select the **By Points** construction method;
- ▼ Select a different coordinate axis;
- ▼ Select the **By Objects** construction method;

Setting offset of the vertex

There are two methods of setting offset for the vertex to be created.

Method 1

Offset of the vertex against another vertex along the construction line is set (**distance** between vertices).

Method 2

Offset of the vertex against the specified face is set. In this case you must select a **base face** and set **offset** from it. The base face may be represented by any face or plane.

In this case, the position of the vertex being created is determined as follows (see Fig. 2.5.29).

1. A surface equidistant to the reference surface and positioned at distance from it.
2. The intersection point for the construction line and equidistant surface is found.
If required, theoretical surface of the face will be prolonged (see Section 2.5.1.3 for more information on the face's theoretical surface).
3. The vertex to be created is mated to the resulting intersection point.

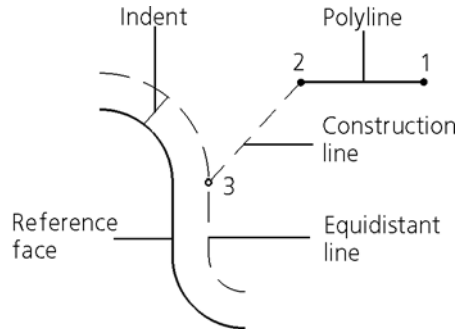


Fig. 2.5.29. Determination of the vertex position by setting its offset against the reference face.



- ▼ To specify the vertex offset using the first method, you should activate the **Offset from the Previous Vertex** switch in the **Offset** group of the Properties bar.

Specify the necessary value in the **Distance** field. Sign preceding the distance value determines the side from the previous vertex on the plotting line on which the current vertex is to be positioned.



- ▼ To specify the vertex offset using the second method, you should activate the **Offset from the Base Surface** switch. The elements for managing vertex offset will appear on the Properties Bar.



To specify a base face, activate the **Base Surface** switch and select the required face in the model window. The selected face is highlighted. The name of the selected object is displayed in a field beside the switch.

Specify the necessary value in the **Offset** field. The sign preceding the offset value determines the side from the reference surface on which the vertex is to be positioned.

The position of the vertex can be set by specifying a point in the model window (see Sections: *Specifying vertex position in the Model window with the Cursor* on p. 383, *Linking the vertex to a point object* on p. 383).

After defining parameters of the offset, press the <Enter> key.

A new vertex will appear in the Model window. Its parameters line will be added to the Vertices Parameters Table.

Specifying vertex position in the Model window with the Cursor

The vertex position of a polyline or spline can be explicitly defined with the cursor in the Model window. In this case the vertex coordinates coincide with the cursor position projection on the vertex construction line.

To build a vertex click in the Model window.

A new vertex will appear in the Model window. Its parameters line will be added to the Vertices Parameters Table.

Linking the vertex to a point object

The vertex of a spline or a polyline created by axes can be linked to a point object. To do this click on the desired point object with the cursor. In this case the curve vertex will coincide with the projection of the selected point object onto the vertex construction line.

To specify a point object hover the cursor over it in the Model window. When the symbolic notation of a line appear next to the cursor, click the left mouse button.

The new vertex will appear in the Model window. The parameters row for the new vertex in the Vertices Parameters Table will be populated automatically. The coordinate value by the selected axis will be the same as the respective coordinate of the selected point object.

If the **Associate** option is on, the vertex is associated with the selected point object (see Section 2.5.4.8.6 on p. 384).

If the curve vertex was created with the **Associate** option off, the curve vertex will coincide with the selected point object (or its projection), but will not be related to it - i.e. while moving the point object the vertex position will remain unchanged.

2.5.4.8.5. Construction by Objects

Construction of a spline or a polyline by objects is limited by the following restriction to the direction of the vertex line construction: To define the line direction, use the existing straight or flat objects in the model window. The vertex construction line may run parallel to a straight object or perpendicular to a straight or flat object.

If the vertex construction line is created parallel to the object, you may employ a vector as a director for such construction line.

A vector may be created with the **Vector drawing** command without leaving the curve construction sequence (see Section 2.8.5 on p. 613).

The following methods can be used to define the vertex coordinates on the construction line:

- ▼ setting offset of the vertex to be created against the previous surface or a preset surface (see Section *Setting offset of the vertex* on p. 381),
- ▼ Specifying the vertex position in the Model window (see Section *Specifying vertex position in the Model window with the Cursor* on p. 383);
- ▼ Linking the vertex to a point object (see Section *Linking the vertex to a point object* on p. 383).

Creating the vertex construction line parallel to the object

The curve vertex construction line may be created parallel to an existing straight object (see Table 2.1.7 on p. 131). Moreover, the object defining the direction of the construction line may

be represented by a vector. This vector can be created with the **Vector drawing** command without leaving the curve construction sequence (see Section 2.8.5 on p. 613).

To create a vertex construction line perform the following operations.

1. From the **Construction Method** drop-down list select the **Parallel to Object** option.
 2. Select the guide object in the window or in the Design Tree or build a vector.
- ▼ To select a guide object in the model window move cursor next to it. When the symbolic notation of the object appears click the left mouse button.
 - ▼ To create a vector defining the direction of the vertex construction line click the **Construction of Vector** on the Advanced Control panel.



Creating the vertex construction line perpendicular to the object

The curve vertex construction line may be created perpendicularly to an existing straight or flat object (see Table 2.1.7 on p. 131).

To create a vertex construction line perform the following operations.

1. From the **Construction Method** drop-down list select the **Perpendicularly to Object** option.
 2. Select the guide object in the window or in the Design Tree.
- To select a guide object in the model window move cursor next to it. When the symbolic notation of the object appears click the left mouse button.



Building a vertex construction line perpendicularly to a straight object passing through the previous vertex of the curve is not possible since the construction direction cannot be identified positively.

2.5.4.8.6. Creating Vertex Associations with Point Objects. Peculiar Features of Associated Vertices

The vertex of a spline or a polyline may contain an associative relation with a point object. Due to the available relation, the vertex will follow any moves of the object. An associative relation is built if the **Associate** option is on during the point object selection. The current vertex row in the Vertices Parameters Table will show a symbol denoting the vertex relation to the point object (see Fig. 2.5.27 on p. 377).

You can use reference points when creating a polyline or spline vertex by any method: **By Points**, **By Axes** and **By Objects**.

- ▼ When constructing “By Points” the vertex directly coincides with the specified point object.
- ▼ When constructing by axes or by objects the vertex coincides with the projection of the specified point object onto its construction line.

In both cases the relation with the point object is maintained until manually deleted. A change in the method of constructing a vertex does not invalidate the relation.

For example, suppose, a vertex related with a reference point was constructed “by points” - i.e. directly coinciding with the reference point. During subsequent editing the user chose to apply “by axes” method of construction to the vertex and specified axis X as the guide. Since the re-

lation with the point object was not deleted, the curve will be reconstructed in such a way that the edited vertex coincides with the projection of the specified point object onto axis X .

Vice versa, if the vertex coincided with the projection of the point object onto the vertex construction line prior to editing, then, as a result of changing the construction method to “by objects”, it will coincide with the point object itself.

If the curve vertex was created with the **Associate** option off, the curve vertex will coincide with the selected point object (or its projection), but will not be related to it - i.e. while moving the point object the vertex position will remain unchanged.

2.5.4.8.7. Closed or open curve



3D splines and polylines may be open or closed. To select a construction variant, activate the **Open Curve** or **Closed Curve** switch in the **Closure** group. When the curve is closed, a segment connecting the starting and the ending vertices is added to it; in case of opening, the segment is deleted.



2.5.4.8.8. Editing Vertex Parameters

You can change the parameters of any already created vertex of a spline or a polyline without leaving the curve construction command.

To edit the existing vertex, select it in the model window or activate the corresponding row in the table of vertices of the curve.

Control elements of the Property Bar allow you to change the method, by which a selected vertex was created: **By Points**, **By Axes** and **By Objects**.

Selecting construction type “by axes” enables changing the axis parallel to which the curve vertex construction line is built.

Selecting construction method “by objects” enables modification or creation of the object used as the guide.

The parameters of a vertex not related to the point object can be changed as follows.

- ▼ In the cells of the vertices parameters table:
 - ▼ set the values of the vertex coordinates;
 - ▼ For a spline: vertex weight;
 - ▼ for a polyline: vertex rounding radius.
- ▼ Modify vertex position in the Model window by the cursor.
- ▼ Change the values of the vertex coordinates or the distance separating the vertex being edited from the previous one using the fields on the Properties bar.

If the vertex is related to a point object its parameters can be edited as follows.

- ▼ Delete the vertex relation to a point object and then use the methods as described above.
- ▼ Modify the point object position.
- ▼ Specify a different point object.



If the vertex was constructed using the **By Points** method with a point object specified, changing the construction method will affect its position.



The table may not be used to associate a vertex with a point object. An associative relation is built if the **Associate** option is on during the point object selection in the Model window. The symbol denoting the relation of the vertex with the point object appears in the row of the current vertex. The relation can be only deleted in the table.



Be careful when deleting a relation, since this action cannot be undone. To restore the relation you should edit the vertex by recreating or specifying the required point.

To switch from editing the spline or polyline vertices and continue to enter vertices, select the **Continue Vertex Input** command from the drop-down menu or activate the last (unnumbered) row in vertices parameters table.

2.5.4.9. 3D Spline



To create a 3D-spline, apply the **Spline** command.

To create a spline, you need to sequentially define vertices of its base polyline (below— “spline vertices”).

The spline vertex can be constructed as follows:

- ▼ **By Points** (see Section 2.5.4.8.3 on p. 379).
- ▼ **By Axes** (see Section 2.5.4.8.4 on p. 381).
- ▼ **By Objects** (see Section 2.5.4.8.5 on p. 383).

Select the vertex construction methods from the **Construction Method** drop-down list on the Properties bar.

The first vertex of a spline can be constructed only using the **By Points** method.

The typical examples of defining a vertex coordinates are:

- ▼ Entering coordinate values into the vertices parameters table (see Section 2.5.4.8.1 on p. 376);
- ▼ creating a special point (see Section 2.5.2.11 on p. 346),
- ▼ Linking a vertex to a point object (see Section *Linking the vertex to a point object* on p. 380).

The **Type** group of switches enables selection of the spline type: spline by vertices or spline by poles (see Section 2.5.4.9.1 on p. 387).

If the original/final spline vertex belongs to a model existing object, you can mate the spline with such object. The **Matings** tab on the Properties bar is used to specify mating conditions (see Section 2.5.4.9.2 on p. 388).

The **Associate** option enables building an associative relation between the spline vertices and point objects (see Section 2.5.4.8.6 on p. 384).

If necessary, you can change the shape of spline by points by controlling tangency and curvature at any of its vertices. To do this, highlight a line in the table of parameters with the number of the required vertex and click the **Control of the form in points** button on the special control bar. The system will enter the spline form control for the current vertex mode (see Section 2.5.4.9.4 on p. 391).





If the initial and the ending vertex of the spline is mated with the model object, you cannot control the spline form for these vertices. Delete the preset mating to be able to control the spline form in this vertex (for example, using the context menu commands **Delete control parameters— In the point** or **Delete control parameters— In all points**).

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the spline, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361). This tab also shows the name of the spline coordinate system name (object coordinate system is detailed in Section 2.8.3.4.1 on p. 598).

The spline with the current parameters is displayed on the screen as a ghost. The ghost dynamically updates on changing parameters.



Upon completion of the point parameterization click the **Create Object** button on the Advanced Control Panel.





The created 3D spline will appear in the model window whereas its icon will be displayed in the Design Tree. Subordinate objects of the spline in the Design Tree are its vertices.

2.5.4.9.1. Spline Type

To select a spline type, enable a desired switch in the **Type** group: **Spline by Vertices** or **Spline by Poles** (Table 2.5.8).

Tab. 2.5.8. Spline types

Curve Type	Construction result
 Spline By Vertices	Spline runs through all vertices with coordinates specified in the table.
 Spline by poles	The spline goes through the extreme vertices tangent to outermost segments of the reference polyline and does not go through intermediate vertices (they are called the spine's "poles"). Distance from the spline to the pole depends on the Weight parameter of this pole.

When building a spline by poles, the **Order** and the **Weight** fields become available.

By default, the spline has order 4, while all poles have the weight of 1. The order may be in the range from 2 through 10 whereas the weight — in the range from 0.0001 through 999.

- ▼ To set the spline's order, enter the right value in the **Order** field.
- ▼ In order to set the weight of the spline's pole, select the pole and enter the required values in the **Weight** field. The pole's weight can also be specified in the **Weight** column of the table of the vertices' parameters (see Fig. 2.5.27 on p. 377, a).

For details on curve order and point weight refer to Section 2.5.1.4 on p. 329.

2.5.4.9.2. Mating Spline with Model Objects

The spline may be mated to an object existing in the model (**mating object**) at the initial or the final vertex (**mating vertex**). In order to perform a mating operation the mating vertex must belong to the mating object.

The following may be used as mating objects:

- ▼ single-segment 3D curves (or separate segments of the multi-segment curves),
- ▼ auxiliary and coordinate surfaces,
- ▼ sketch lines,
- ▼ edges,
- ▼ faces of surfaces and bodies

The user may select one of the four mating conditions: **Not Specified**, **By Tangent**, **Smooth** and **Perpendicularly** (see Section 2.5.1.7 on p. 332).

The mating conditions are specified during creation or editing the spline on the **Matings** tab of the Properties bar.

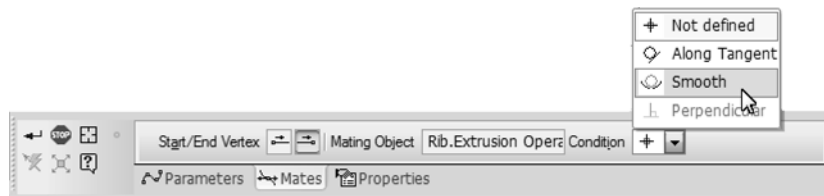


Fig. 2.5.30. **Matings** Tab

Availability of each mating condition for the initial or the final vertex depends on the mating type.

- ▼ If the mating subject is represented by a surface or a face, the following mating conditions are available: **Not Specified** and **Perpendicularly**.
If the mating vertex lies on the curve being part of a surface or a face (e.g. on an edge), the curve can be used as the boundary object in relation to which the mating condition is specified. In this case all the mating conditions will be available.
- ▼ If the mating subject is represented by a 3D line being a straight object (i.e. polyline segment), the following mating conditions are available: **Not Specified**, **By Tangent**, and **Smooth**. Mating with other types of 3D curves can be performed under all the mating conditions (types of 3D curves are listed in Table 2.1.7 on p. 131).

The mating direction can be changed using the **Direction** group of switches. The group becomes unavailable on selecting a spline mating condition — **Not specified**.

Mating

To mate a spline to a mating subject:

1. Select the initial or the final vertex of the spline to specify mating parameters. To do this, activate the desired switch in the **Initial/Final Vertex** group. The selected vertex will be highlighted in the model window.
2. Select the mating subject in the model window. The selected object will be highlighted whereas its name will appear in the **Mating Subject** field.
3. If necessary, specify the boundary object in the model window. The selected object will be highlighted whereas its name will appear in the **Boundary** field.
4. Select a mating condition from the **Condition** list. The conditions which may not be applied to the vertex selected are not available.

The ghost of the spline being created in the model window will be reconstructed according to the mating condition selected and the mating condition symbol will appear next to the vertex.



Spline mating **By poles** with other objects have some peculiarities, see Section 2.5.4.9.3 on p. 389.

Deleting a Mating

The user may delete the spline mating at the initial or the final vertex or at all the vertices simultaneously.

To delete the specified mating at the current — i.e. initial or final — vertex use the **Delete Control Parameters— At Point** drop-down menu command. To delete the specified matings at all the vertices use the **Delete Control Parameters— At All Points** drop-down menu command.



The specified mating may be deleted at the initial or the final vertex by means of the **Reselect** command. To do so activate the necessary switch in the **Initial/Final Vertex** group of switches on the Properties Bar, then click the **Reselect** button on the Advanced Control panel.

Please be advised that you may not restore the mating once deleted. You may only redo the mating in order to restore it.

If the spline vertex is assigned the system pole attribute, the spline mating may be deleted by removing the attribute. To do so highlight the system pole row in the vertices parameters table and click on the system pole icon in the **Relation** column or use the **Delete Relation** command from the table drop-down menu. The relation will be deleted. The vertex coordinates will remain unchanged.

2.5.4.9.3. Peculiarities of the spline's mating **By poles** with Other Objects. System pole

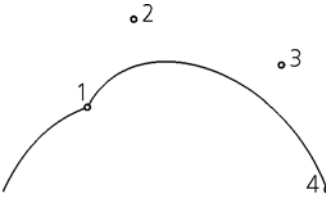
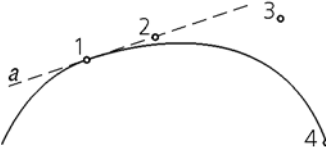
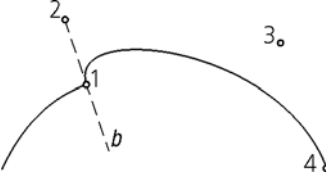
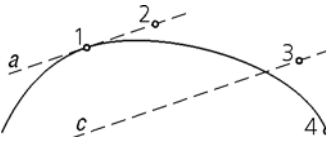
Using **Perpendicular**, **By tangent** and **Flat** mating conditions for a spline **By poles** has the following peculiarities.

The mating is formed by applying certain geometrical limitations in the position of the spline's poles adjacent to the mating vertex. These constraints are automatically applied to poles after the mating condition selection. If you need, a pole is offset by the minimum required distance. The spline's poles with automatically applied limitations become **system poles**. In the Vertices

Coordinates table system poles are marked with a special sign, see Fig. 2.5.27 on p. 377.

Limitations applied to system poles and the number of the system poles required for meeting different conditions of the mating are listed in Table 2.5.9.

Tab. 2.5.9. Number of system poles required for mating conditions, and geometrical constraints for these poles

Mating condition	Number of system poles; geometrical constraints for them	Example*
Not specified	0; no constraints	
By Tangent	1; constraint: relation of the straight line <i>a</i> , tangent to the mating object in the mating point	
Perpendicular	1; constraint: relation of the straight line <i>b</i> , perpendicular to the mating object in the mating point	
Smooth	2; constraint for the first pole: relation of the straight line <i>a</i> ; constraint for the second pole: relation of the straight line <i>c</i> parallel to <i>a</i> at a distance depending on the mating object's curvature in the mating point and on the distance between this point and the first pole.	

* In Figures as numbers are marked: 1 – the vertex of the spline's mating with the object, starting vertex; 2 and 3 – poles of the spline; 4 – ending vertex of the spline.

Spline's mating in the initial (ending) point is possible, if it has the number of adjacent poles sufficient for the mating; these poles should not be system ones (i.e. they should not be used for mating in the opposite vertex).



For **Flat** mating, the order of the spline should be 4 or higher.

If the poles adjacent to the mating vertex are associated with point object or constructed by means of the **By Axes** or **By Objects** method, upon the mating condition selection the message regarding changing the construction method and re-positioning of vertices will appear on the screen.

Click **OK** to confirm modification of the vertex parameters: i.e. change of the construction method to **By Points**, cancellation of the associative relation with the point object and, if necessary, change of coordinates. The edited vertex acquires the status of a system pole — this will be indicated by the system pole symbol in the vertices parameters table in the corresponding row of the vertex (see Fig. 2.5.27 on p. 377).

To cancel modification of the vertex parameters, click the **Cancel** button in the message. In this case the mating condition remains unchanged — i.e. **Not Specified**.

2.5.4.9.4. Handling the Spline Shape

If the spline is built by points, you can change its form managing tangency and/or curvature in the current vertex.



Changing the spline's form is possible in the spline's form management mode for the current vertex. To enter this mode, select the required vertex in the model window, or the line with this vertex's number in the parameter table, and then click the **Spline form management in vertices** button on the Special control bar.



If the initial and the ending vertex of the spline is mated with the model object, you cannot control the spline form for these vertices. To control the spline form in the starting or ending vertex, delete the preset mating in this point (for example, using commands of the contextual menu **Delete control parameters— In the point** or **Delete control parameters— In all points**).

After entering the management mode, the **Parameters** tab contains the table of the vertices' parameters and the **Control** list with options of the spline form control for the current vertex (Fig. 2.5.31). In the list, select the desired control option.

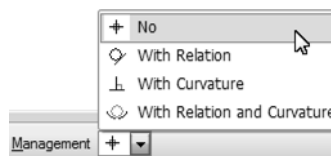


Fig. 2.5.31. **Control List**

The control options are described in more detail below.

Set parameters of the selected option using control elements of the Property bar or using the control ghost in the model window.

The ghost of the spline to be created in the model window will be re-built, a symbol of the control option appears near the current vertex.

In the spline form control mode, you can change the position of the current vertex by moving it with the mouse to the model window or by setting the required values of the vertex's coordinates in the parameter table (see Section *Changing of the vertex parameters in the table of parameters* on p. 378).

After finishing editing the spline's form at one of vertices, you can:

- ▼ set the spline's form for its another vertex; to do this, highlight the required vertex in the model window or a line with this vertex number in the vertices' parameter table,
- ▼ leave the spline's form control mode; to do this, click the **Form control in points** button on the Special control bar,
- ▼ finish construction of the spline — to do this, click the **Create Object** button.



Tangency control

Use this option to control the spline's form for the current vertex by setting the tangent's direction and the value of tangency.

Value of tangency indicates how close is the spline's form to the tangent circle near its current vertex (see Section 2.5.1.6 on p. 331 for more information about the tangent circle).

Curvature of the spline in the current vertex is defined by the system automatically using the preset tangency value.

To use this option, choose the **Tangency** option in the **Control** list of the Property bar.

Using the property bar's control elements or hot points of the controlling ghost, set the tangent's direction for the current vertex and the tangency value for this vertex. For more information on setting parameters listed above see Section 2.5.4.9.5 on p. 393.

Curvature control

Curvature control is control of the spline form on a given vertex by setting curvature or the radius of curvature.

- ▼ Setup of curvature includes determining of normal vector direction and specifying of curvature value.
- ▼ Setup of curvature radius includes determining of curvature center location and specifying of curvature radius value.

The spline's tangency at the current vertex is defined by preset curvature value and curvature radius value.

To use this option, choose the **Curvature** option in the **Control** list of the Property bar.



Using control elements on the Properties toolbar or handles of the managing ghost, specify direction, curvature or curvature radius in the current vertex. For more information on setting parameters listed above see Section 2.5.4.9.6 on p. 394.

Tangency and curvature control

Use this option to change the spline's form at the current vertex using two parameters—tangency value and curvature.

To use this option, choose the **Tangency and curvature** option in the **Control** list of the Property bar.

Select a method of control using switches of the **Method** group:

-  ▼ Enable the **Tangent orientation and tangency value control** switch to set the direction of tangent and tangency value at the current vertex (see Section 2.5.4.9.5 on p. 393).
-  ▼ Enable the **Normal and curvature orientation control** to change the spline's form managing the curvature or curvature radius (see Section 2.5.4.9.6 on p. 394).

Aside from enabling the **Method** group of switches, you can click the mouse at hot points of the “Tangency” ghost “Tangency” and “Curvature” to change between the methods.

2.5.4.9.5. Setting orientation of the tangent and the tangency's value

You can set the orientation of the tangent and tangency value using the controls of the Properties bar (Fig. 2.5.32) or using the handles of the managing ghost (Fig. 2.5.33).

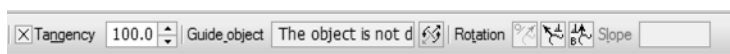


Fig. 2.5.32. Property bar when controlling the tangent's orientation and tangency's value

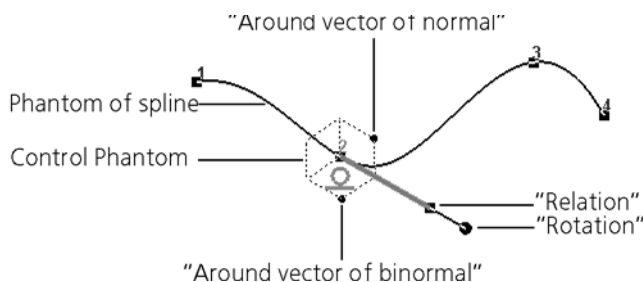


Fig. 2.5.33. Controlling ghost when controlling tangent's orientation and tangency

Direction of the tangent vector in the current vertex

You can set the direction of the tangent vector in the current vertex using the guide object or the rotation of the managing ghost.

Setting of the guide object for the tangent vector is detailed in Section 2.5.4.9.7 on p. 396, rotation of the managing ghost— in Section 2.5.4.9.8 on p. 396.



If required, you can reverse the obtained direction of the tangent vector. To do this, click the **Change Direction** switch.

Tangency

To set up the spline's tangency value at the current vertex, enter the required value into the **Tangency** field and press the <Enter> key. The spline ghost is being reconstructed.

You can also set up tangency value using the handle of the “Tangency” controlling ghost (see Fig. 2.5.33 on p. 393).

2.5.4.9.6. Setup of the Curvature and the Orientation of the Normal

You can set the curvature or the radius of curvature using the controls of the Properties bar (Fig. 2.5.34) or using the handles of the managing ghost (Fig. 2.5.35 and 2.5.37).

Setup of Curvature

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To set up the curvature in the current vertex, activate the **Curvature** switch in the **Parameter** group. Curvature controls will become available on the Property bar (see Fig. 2.5.34).

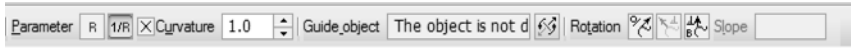


Fig. 2.5.34. Property bar when controlling normal orientation and curvature
The managing ghost will be displayed (Fig. 2.5.35).

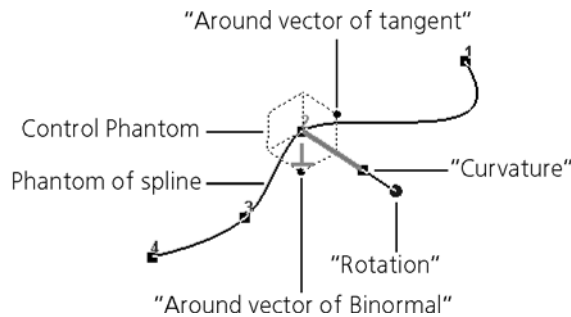


Fig. 2.5.35. Managing ghost when specifying the curvature

Setup of curvature includes determining of normal vector direction and specifying of curvature value.

Direction of the normal vector in the current vertex

To determine the normal vector direction, use the following methods:

- ▼ setting of the guide object;
- ▼ rotation of the managing ghost,

For more information on setting the normal vector direction see Section 2.5.4.9.7 on p. 396, on rotation of the managing ghost— see Section 2.5.4.9.8 on p. 396.



If required, you can reverse the direction of the normal vector. To do this, click the **Change Direction** switch.

Curvature

To set up the curvature value, enter the desired value into the **Curvature** field and press the <Enter> key. The spline ghost is being reconstructed.

You can set up value of curvature using the handle of the “Curvature” managing ghost (see Fig. 2.5.35).

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Specifying the curvature radius

To set up the curvature radius in the current vertex, activate the **Curvature Radius** switch in the **Parameter** group.

Control elements for joining curves will appear on the Properties Bar (Fig. 2.5.36).

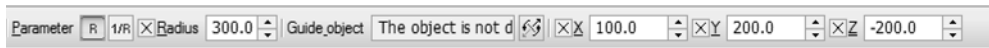


Fig. 2.5.36. Property bar when controlling the center's position and curvature radius value
The managing ghost will be displayed (Fig. 2.5.37).

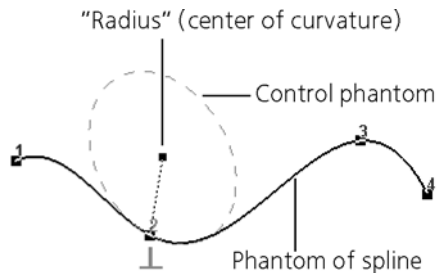


Fig. 2.5.37. Managing ghost when specifying the curvature radius

Setup of curvature radius includes determining of curvature center location and specifying of curvature radius value.

Radius of curvature may be specified using one of the following methods:

▼ Manual specifying of the center of curvature

Enter the desired coordinates of the center of curvature into the corresponding fields of the Properties bar. The controlling ghost assumes the desired position. The distance from the current vertex to the center of curvature is displayed in the **Radius** field. If needed, you can change the obtained value of the radius of curvature. To do this, enter the desired value into the field and press the <Enter> key.

The controlling ghost will be re-built – the center of the curvature will move at the desired distance along the radius. Coordinates of the center of curvature are recalculated.

In addition, position of the center of curvature can be set with mouse in the model window. To do this, move the handle of the managing ghost to the required position. Note that the tangent circle and normal vector rotate around the tangent vector.

▼ Use of Guide Object

This method includes the following actions:

- ▼ specifying a guide object for the radius of curvature in the current vertex,
- ▼ specifying of the radius of curvature value at that vertex.

Any point, straight or planar object can be used as a guide object (see Table 2.1.7 on p. 131).

The direction defined by a point object is a line passing through this point object and the current vertex, by a straight object—a line, parallel to the object, by a flat object—a line, perpendicular to the object. Specify the guiding object in the Design Tree or Model Window.



Moreover, the guide object may be represented by a vector. To create a vector, click the **Vector drawing** button on the Advanced Control Panel (see Section 2.8.5 on p. 613).

The name of the selected or created object is displayed in the **Guide Object**. The controlling ghost assumes the preset position, the curvature center is positioned on the radius, the radius will have its current value.

Specify the radius of curvature value. To do this, enter the desired value into the **Radius** field and press the <Enter> key. The spline ghost is being reconstructed.



Note that if a point objects is used as a leading object, the curvature center will coincide with this object. The distance from the current vertex to the center of curvature is displayed in the **Radius** field, the fields for entering the coordinates of the center of curvature are not available.

You can also set up value of the radius of curvature using the handle of the “Radius” managing ghost (see Fig. 2.5.37).



You can reverse the obtained (by the first or the second method) direction of the radius of curvature. To do this, click the **Change Direction** switch of the Properties bar.

2.5.4.9.7. Guide object for the vector

Any point, straight or planar object can be used as a guide object for the tangent vector or the normal vector (see Table 2.1.7 on p. 131).

The direction defined by a point object is a line passing through this point object and the current vertex, by a straight object—a line, parallel to the object, by a flat object—a line, perpendicular to the object.



In addition, the guiding object may be represented by a vector (see Section 2.8.5 on p. 613). To create a vector, click the **Create Vector** button on the Advanced control panel.

Select the guide object in the Design Tree or in the model window or build a vector.



The name of the selected or created object is displayed in the **Guide Object**. The controlling ghost assumes the desired position, the spline's ghost will be re-built.



If a straight line object is used as a leading object, you can select vector's direction: parallel or perpendicular to the guide object. This is done by the **Orientation** group of switches.

2.5.4.9.8. Rotation of the managing ghost

You can rotate the managing ghost around any of the basis vectors using the controls of the Properties bar (on the basis vectors, see Section 2.5.1.5 on p. 331).

The **Rotation** group of switches of the Properties bar allows to select a vector around which the managing ghost will rotate.



- ▼ Activate the **Around the Tangent Vector** switch so that the ghost rotates around the tangent vector. After that, enter the angle of revolution value into the **Angle** field.



- ▼ Activate the **Around the Normal Vector** switch so that the ghost would rotate around the normal vector. After that, enter the angle of revolution value into the **Angle** field.



- ▼ Activate the **Around the Binormal Vector** switch so that the ghost would rotate around the binormal vector. After that, enter the angle of revolution value into the **Angle** field.

The controlling ghost assumes the desired position, the spline's ghost will be re-built.

You can also rotate the guide ghost around the basis vectors using the handles “Around the Tangent Vector”, “Around the Normal Vector”, “Around the Binormal Vector” or, arbitrarily, us-

ing the “Rotation” handle (see Fig. 2.5.33 on p. 393 and 2.5.35 on p. 394).

- ▼ If a **Tangency** control option is used, the rotation of the controlling ghost is only possible around the normal vector or binormal vector. That is why ghost does not have “Around the Tangent Vector” handle and **Around the Tangent Vector** switch is unavailable in the **Rotation** group.
- ▼ If a **Curvature** control option is used, the rotation of the controlling ghost is only possible around the tangent vector or binormal vector. That is why ghost does not have “Around the Normal Vector” handle and **Around the Normal Vector** switch is unavailable in the **Rotation** group.
- ▼ If a **Tangency and curvature** control option is used, the rotation of the controlling ghost is possible around all basic vectors. Therefore, the ghost contains all handles and all switches in the **Rotation** group are accessible.

Note that if a tangent or a normal relates to the leading object when using the **Tangency and curvature** option, its position in the space will be fixed. In this case, you can set direction of another straight line (normal or tangent):

- ▼ by rotation of the controlling ghost around the desired straight line using its hot point,
- ▼ by specifying the leading object which is perpendicular to the desired straight line.

2.5.4.10. 3D Polyline



To construct a 3D polyline, apply command **Polyline**.

To create a polyline, you need to sequentially define its vertices.

A polyline vertex can be constructed as follows:

- ▼ **By points** (see section 2.5.4.8.3 on p. 379);
- ▼ **By Axes** (see Section 2.5.4.8.4 on p. 381);
- ▼ **By Objects** (see Section 2.5.4.8.5 on p. 383).

Construction method can be selected from drop-down menu **Construction Method**.

The typical examples of defining a vertex coordinates are:

- ▼ Picking a vertex position by the cursor in the Model window,
- ▼ Entering coordinate values into the vertices parameters table (see Section 2.5.4.8.1 on p. 376);
- ▼ creating a special point (see Section 2.5.2.11 on p. 346),
- ▼ Linking a vertex to a point object (see Section *Linking the vertex to a point object* on p. 380).



When picking a vertex position by the mouse in the Model window, take into account the screen plane position in the three-dimensional space of the model window.

The first vertex of a polyline can be constructed only **By points**.

Second vertex is created **By Axes** by default. You can select another mode of construction for this vertex.

When constructing the a new vertex by any method its construction line connecting the vertex with the previous one will appear on the screen. This lets you judge correctness of the input parameters and edit them if necessary.

If necessary, the user may construct several vertices of a polyline between two existing vertices (see Section 2.5.4.10.1 on p. 398). When the construction is finished the new vertices are added to the parameters table. The polyline vertices are re-numbered. To resume construction of new vertices, select the last unnumbered row in the table.

The **Associate** option enables building an associative relation between the polyline vertices and point objects (see Section 2.5.4.8.6 on p. 384).

You can round the polyline's vertices, see Section 2.5.4.10.2 on p. 398.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the polyline, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361). This tab also shows the name of the polyline coordinate system name (object coordinate system is detailed in Section 2.8.3.4.1 on p. 598).

The polyline with the current parameters is displayed on the screen as a ghost. The ghost dynamically updates on changing parameters.



Upon completion of the point parameterization click the **Create Object** button on the Advanced Control Panel.



The created 3D polyline will appear in the model window whereas its icon will show in the Design Tree. Subordinate objects of the polyline in the Design Tree are its vertices.

2.5.4.10.1. Inserting of Range of Vertices



To insert range of vertices between the two current ones you should click the **Insert range of vertices** button. The system initiates creation of new vertices between two existing curve vertices.

Use the cursor to define boundary curve vertices that will limit the set being inserted.



A vertex cannot be selected as a second vertex if it has the number which is less than the number of the first one.

The curve section between boundary vertices will disappear.

Create new vertices of the polyline. The methods of building a range of vertices are identical to those used for constructing a polyline (see Section 2.5.4.10 on p. 397). The parameters of the inserted vertices are displayed in the vertices parameters table.



To complete insertion click the **Create Object** button on the Special Control Panel.

The vertices of the inserted set will replace the vertices that previously were between the boundary ones. The inserted vertices' parameters will be added to the Parameters Table. The polyline vertices will be renumbered.

2.5.4.10.2. Vertex rounding

In order to round a polyline's vertex, highlight the required vertex in the model window and enter the value of the rounding radius in the **Radius** field in the Property Bar. Besides that, you can enter radius in the respective cell of the table of the vertices' parameters (see Fig. 2.5.27, b).

If the polyline is closed, you can round any vertex. If it is open, any vertex can be rounded, except for the starting and ending ones.

You can also control the rounding radius in the model window by moving handles. The **Handles** group of switches in the Property bar allows you to specify, which handles of a polyline should be displayed on the screen:



- ▼ If the **All handles** switch is enabled, two points will be displayed in each vertex (except for the starting and the ending vertices of the open polyline): one for changing the vertex's position; another one for changing the rounding radius.



- ▼ If the **Handles of vertices and non-zero fillets** switch is enabled, the point for changing the fillet radius is displayed only if the value of the radius is not equal to zero. Thus, in the vertices with zero radius (as well as in the starting and ending vertex of the open polyline) one point is displayed; in all other vertices, two points are displayed.

2.5.4.11. Spline by Objects

You can create a spline with the shape identical to the shape of a model object (or objects). You can use 3D curves, edges of bodies and surfaces, contours in sketch as source objects.

- ▼ A single spline shall be created when source objects form a chain and are smoothly connected.
- ▼ Several separate splines shall be created when source objects do not form a chain or are not smoothly connected.



A spline created by source objects does not differ from a spline created by the **Spline** command (see Section 2.5.4.9 on p. 386). That is why you can use standard methods of spline editing for the constructed spline.



To create a 3D spline by source objects, call the **Spline by Objects** command.

Select an object or objects for creation of a spline in the model window. 3D lines may be defined in the Design Tree.



Source objects may be also selected before the command selection.



To cancel object selection, re-select it.

The switches of the **Mode** group allow you to select whether the source objects will be saved after the application of the command or not.



The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the spline, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361). This tab also shows the name of the spline coordinate system name (object coordinate system is detailed in Section 2.8.3.4.1 on p. 598).



To finalize the construction, click the **Create Object** button on the Special Control Panel.



- ▼ If you select spline construction without deletion of source objects, then the created spline will appear in a model window and the spline's icon will appear in the Design Tree.
- ▼ If you have selected the spline construction mode with deletion of source objects then the object deletion dialog should be displayed. The dialog allows to preview the model objects being deleted. You can cancel or confirm the deletion.

- ▼ If deletion is canceled: the spline appears in the model window, while initial objects remain.
- ▼ If deletion is confirmed: a spline appears in the model window, while initial objects disappear.



Note that no link is formed between the spline and the selected object, i.e. deleting or editing the object does not affect the created spline.



Only objects that belong to it, but not to its components, can be deleted from a model. It is impossible to delete the objects that belong to the assembly components. If you select the objects of components for spline creation, then, after clicking the **Create Object** button, a spline will be created and the screen will show a message that the deletion of selected objects is impossible.

2.5.4.12. Curve by law

Allows to create a curve by specifying it with parametric equations.

For details on curve parametric representation see Section 2.5.1.2 on p. 327.

Curve is an object that uses coordinate system (for object coordinate system, refer to Section 2.8.3.4.1 on p. 598).

Parametric equation of the curve can be set in the rectangular, cylindrical or spherical coordinates. By default, the curve is constructing in the rectangular coordinate system that coincides with the curve coordinate system. Cylindrical or spherical coordinate systems are relative to the coordinate system of a curve in the same way as of a point (see Section 2.5.3.4.1 on p. 358).

To create a curve by parametric equation, complete the following steps.



1. Call the **Curve by Law** command.

The curve ghost will be displayed in the default view.

2. Select a coordinate type to be used for curve construction. To do this, activate the relevant switch in the **Coordinate Type** group on the Properties bar:



- ▼ **Rectangular X,Y,Z**,



- ▼ **Cylindrical R,A,Z**,



- ▼ **Spherical R,A,B**

3. Specify functions for the point coordinates of the curve (see Section 2.5.4.12.1 on p. 401).

All changes are displayed in the curve ghost in the model window.

4. If necessary, change name and color of the created curve, set up its display parameters. For that, use control elements on the **Properties** tab of the Property bar. This tab also displays the name of the curve coordinate system.

For detailed information on display parameters of the curve see Section 2.5.4.1 on p. 361.



5. Click the **Create Object** button on the Special Control Panel.

The created curve appears in the model window, while its icon appears in the Design Tree.

2.5.4.12.1. Setup of function for the point coordinates of the curve

You can set function for each coordinate of the curve's point by a linear, cubic, or parametric equation (see Section 2.5.1.2 on p. 327 for more information about the curve's parametric representation). Moreover, you can set a constant value for a coordinate.

The set of coordinates, for which functions are set, depends on which type of coordinates had been chosen for construction of the curve (see p.2, Section 2.5.4.12 on p. 400). For example, if the curve is being constructed in the cylindrical coordinate system, functions are set for coordinates R, A, Z.

Setting a function for the coordinate includes selection of the function's type, and then setting its parameters.

To select a function type for the coordinate, open the **Function by coordinate name** list on the property bar and specify the required line in it (see Fig. 2.5.38).

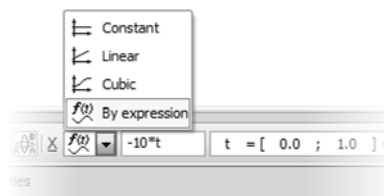


Fig. 2.5.38. **Function type by X** list

The Property Bar will display controls for setting up the parameters of the function of the selected type. Setting parameters of the function of each type is described in Table 2.5.10.

Before completion of the command the system “remembers” the data entered for every type of function, so you can switch between function types without loss of their data.

Tab. 2.5.10. Setup of function parameters for the point coordinates of the curve

Type of function	Setup of function parameters for the point coordinates of the curve
Constant	The coordinate is set by a constant value. Enter or set by counter the required value of the coordinate in the Value by the coordinate name field.
Linear	The coordinate's function is set by a linear equation. Enter the start and end coordinate value in the respective fields on the Property bar.
Cubic	The coordinate's function is set by a cubic equation. Enter the start and end coordinate value in the respective fields on the Property bar

Tab. 2.5.10. Setup of function parameters for the point coordinates of the curve

Type of function	Setup of function parameters for the point coordinates of the curve
By expression	<p>The coordinate's function is set by a parametric equation. Enter the functions' equation, which determines the coordinate, in the Expression of the function by the coordinate name field. Then, enter data about the function's parameters in the Interval parameter field: name and interval to change.</p> <p>More information on entering the function's expression is described in Section 2.5.4.12.2. Description of the parameter's function is described in Section 2.5.4.12.3 on p. 405.</p>



You can use constants to set the point's coordinates (see Table 4 Appendices IV). For functions of the **Constant**, **Linear**, **Cubic** type, the designation of the constant in the coordinate field will be replaced by its value after entering the constant and pressing the *<Enter>* key. For functions of the **By expression** type, the value of the entered constant will be used during the calculation of the coordinate by the defined function's expression.



A curve can not be created if for all point coordinates the **Constant** type of function is selected. In this case, a warning appears on the screen that at least one coordinate function should not be a constant.

2.5.4.12.2. Entering an expression and setup of function parameters

To enter the function's expression, which defines a coordinate, use the **Function's expression by the coordinate name** field.

The function's expression may contain an **interval** parameter and **non-interval** parameters (see Section 2.5.4.12.3 on p. 405). For example, in the function's expression for Z coordinate which is shown on Fig. 2.5.39, t parameter is an interval one, while parameters r and h are non-interval ones.



Note that the name of the interval parameter should not be the same as the names of non-interval parameters.

The following methods can be used to enter the expression:

- ▼ Manual entering of expression,
- ▼ insert a variable model into expression;
- ▼ insert expression elements from collection;
- ▼ insert group law from collection—insertion of function's expressions for all coordinates at once.

Examples of expression entering are described below.



If there is no interval parameter function in the expression, the function's value will be the result of this expression's calculation, i.e. the constant. To make creation of a curve possible, a function of at least one coordinate should contain an interval parameter, i.e. it should not be a constant.

An expression can include any number of blank spaces. Those are ignored when interpreting the expression.

After entering an expression, press *<Enter>*.



If there are Russian letters in an expression or operation signs do not match, or there are symbols which do not denote operations/functions, the syntax error warning message will appear on the screen.

After entering the function's expression, non-interval parameters are inserted into the **Parameters' values** table on the property bar.

Table consists of three columns. The first column represents the names of expression parameters, the second— expressions for their calculation, the third— their values (see Fig. 2.5.39). Work with table is similar to work in Variables Window (for details about Variables Window see Section 7.1.1.3 on p. 1571).

Name	Expre...	Value
r	20.0	20.0
h	5.0	5.0

Fig. 2.5.39. Example of the table of function parameter values along Z coordinate

Manual entry

You can enter the function's expression in the **Function's expression by the coordinate's name field** from the keyboard.

When entering expressions, you can use existing user variable model as parameters, including functional models (see Section 7.1.5 on p. 1589). For this, parameter name should match the variable name. The data on variable (for a user variable – name and expression, for a functional variable – name, expression and argument) is imported into a **Values of the Parameters** table of the Variables Window. Changing of this data in table is unavailable.

If the name of parameter does not match with any of the variables names, this parameter is recorded into table with default value. Activate the cell of parameter by double clicking the mouse in the **Expression** column and enter the desired parameter value. Press the *<Enter>* key. Once the curve has been created, the variable with the name of the preset parameter will appear in the list of the model's variables.



The functional variable which is used in the function's expression should be created in the Variables window prior to creation of a curve by law.



After creation of a curve, you can edit variables used in it— enter a new value of variable or an expression to calculate it in the Variables Window. After the model is rebuilt, the shape of curve is changed.

In expression you can use arithmetical operators, constants, functions and formulas (see Tables of Appendix III and Tables 1, 2, 3 and 4 of Appendix IV). Operators, constants, functions, and formulas can be copied into the **Function's expression by the coordinate's name** via the exchange buffer.

A model variable insertion



You can insert existing variable into the function expression by choosing it from the model variables. To do this, click the **Insert Variable Name** button on the Advanced Control panel. The **Variables** dialog containing the list of all model variables appears on the screen. To specify the required variable, select any cell in a row of this variable and then click the **OK** button in the dialog.

A variable name is added to the expression of function, its data transferred into the **Values of the parameters** table (see Fig. 2.5.39).

The **Insert variable name** button is available, if the cursor is in the **Function expression by coordinate name** field.

insert expression elements from collection



In the function's expression, you can insert element's expressions from collection (system functions, curve function expressions, and so on). To do this, click the **Insert Function from collection** button on the Advanced Control panel. The dialog for insertion of a mathematical expression will appear on the screen. Select the required expression element or make expression from several elements and click the **OK** dialog button. For details on working in dialog, see Section 7.1.3.4 on p. 1582.

The **Insert function from collection** button is available, if the cursor is in the **Function expression by coordinate name** field.

Insertion of a group law from collection

You can set a group law from collection for a curve. In this case every coordinate will be determined by its own function.



To set the group law, click the **Group Law** button of the Special Control Panel. The dialog for insertion of a mathematical expression will appear on the screen. Select the desired group law from the list and click the **OK** dialog button. For details on working in dialog, see Section 7.1.3.4 on p. 1582.

Once the group law is inserted, functions for respective coordinates appear in **Function expression by coordinate name** fields.



If the group law sets functions for X and Y coordinates only (for example, the involute of circle law), the expression of function for Z coordinate will not be changed – default value will be kept. If necessary, you can edit this expression.

2.5.4.12.3. Setting interval parameter of the function

By default, the name of the function's interval parameter is t , interval of its change is $[0.0;1.0]$. You can change the name of the function's parameter and the borders of its interval. For this purpose, enter the required data in the **Interval Parameter** field and press $\langle Enter \rangle$.

The entered data will be added to the list and saved for the duration of a KOMPAS-3D working session.



Note that the name of the function's interval parameter should not be the same as the names of non-interval parameters of the function and the names of the model's user variables.

As threshold values of the interval, you can use numeric values or constants (see Table 4 Appendices IV). After entering a constant and pressing $\langle Enter \rangle$, the designation of the constant will be replaced by its value.

If the function's parameter name for the current coordinate is the same as the name of the function's parameter for a previously set coordinate, the change interval of the function's parameter for the current coordinate will be determined automatically– will be the same as previously set interval. In this case, editing the parameter change interval for the current coordinate will not be available.



Coordinates of a curve's point can be set by functions of different parameters. Each parameter can have its own change interval.

In order to set the change interval for the function's parameter, you can use the existing interval variable (see Section 7.1.5.1.2 on p. 1591). Enter its name in brackets after the equal sign in the right part of the **Interval Parameter** field and then press $\langle Enter \rangle$. Value of the interval variable (change interval borders) is assigned to the operation's parameter. In this case, editing the parameter change interval will not be available.

You can also set the change interval for the function's parameter by specifying a new interval variable with the required value. For this purpose, complete the following steps in the **Interval Parameter** field.

1. In brackets, enter the required interval of values (for example, $[10;-10]$).
2. After the equal sign in the right part of the field, enter the name of the new interval variable (in brackets).
3. Press $\langle Enter \rangle$.

Once the curve is created, the name and the value of the new interval variable will appear in the Variables Window, section **Intervals**.

Henceforth, changing the interval variable value in the Variables Window will be transferred to the curve, and the curve will be rebuilt in accordance with the new interval of the function's parameter change.

When creating a curve by law, interval variables will be created. The values of these variables are the defined intervals of functions' parameters change. They are placed in the variables section of the created curve (see Fig. 7.1.9 on p. 1590).

You can assign the value of an existing user interval variable to the interval variables of the operation's parameter. For this purpose, enter the name of the required user variable in the **Expression** field of the operation's parameter variable. The value of the user variable will be assigned to the operation's parameter variable and transferred to the model, i.e. the parameter change interval borders of the function, that defines the point's coordinate, will be changed.



For the curve to change in accordance with the new interval limits, the model should be rebuilt.

Hence, when changing the value of the user variable, the value of the respective operation's parameter variable will be changed automatically.

Working with variables is described in detail in Section 7.1 on p. 1569.

2.5.4.12.4. Creation of curves by parametric equation – Example

Example 1

You need to construct a parabolic curve set by law: $y = x^2$, within range between -10 and 10.

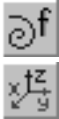
Parametric representation of parabola in the rectangular coordinate system:

$$x = t,$$

$$y = t^2,$$

$$z = 0.$$

We recommend to do the following to create a parabola.



1. Call the **Curve by** command.
By default, the **Rectangular X,Y,Z** switch is enabled in the **Coordinate Type** group. It means that the curve is constructing in the rectangular coordinates.
2. Specify function for the X coordinate.
 - 2.1. Choose the **By expression** function type in the **Function by X** list.
Two fields appear on the Properties bar: the field for entering function's expression and the field for entering the function's parameter name and its change interval. The fields contain default data. name t and default change interval [0;1].
 - 2.2. Enter the parameter change interval in the **Interval parameter** field
 $t = [10;-10]$. Press <Enter>.
3. Specify function for the Y coordinate.
 - 3.1. Choose the **By expression** function type in the **Function by Y** list.
 - 3.2. Enter the function t^2 expression in the *Function expression by Y* field.
The t parameter change interval is defined automatically and is the same as the previously set interval for X coordinate.
 - 3.3. Press <Enter>.
4. Specify function for the Z coordinate.

- 4.1. Choose the **Constant** function type in the **Function by Z** list.
- 4.2. Enter value **0** in the *Value by Z* field.
5. Click the **Create Object** button on the Advanced Control Panel.

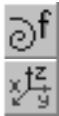
Example 2

For a calculation related to a cable or wire dip, a chain line set by law should be constructed: $y = a \cdot \text{ch}(x/a) = a/2 \cdot (e^{x/a} + e^{-x/a})$, within the range between -50 and 50 if $a = 80$.

Parametric representation of curve in the rectangular coordinate system:

$$\begin{aligned} x &= t, \\ y &= a/2 \cdot (\exp^{t/a} + \exp^{-t/a}), \\ z &= 0. \end{aligned}$$

We recommend to do the following to create a chain curve.



1. Call the **Curve by** command.
By default, the **Rectangular X,Y,Z** switch is enabled in the **Coordinate Type** group. It means that the curve is constructing in the rectangular coordinates.
2. Specify function for the X coordinate.
 - 2.1. Choose the **Linear** function type in the **Function by X** list. Fields will appear on the Properties bar for entering the start and end X coordinate value.
 - 2.2. Enter value **-50** in the *Start Value by X* field.
 - 2.3. Enter value **50** in the *End Value by X* field.
3. Specify function for the Y coordinate.
 - 3.1. Choose the **By expression** function type in the **Function by Y** list. Two fields appear on the Properties bar: the field for entering function's expression and the field for entering the function's parameter name and its change interval.
 - 3.2. Enter the function expression in the **Function expression by Y** field.
 $a/2 * (\exp(t/a) + \exp(-t/a))$.
 - 3.3. Press <Enter>. The **Values of the parameters** table will appear on the Property Bar. Parameter a is entered in it.
 - 3.4. Set the value of the non-interval parameter a . To do this, activate the cell of the **Expression** table by the mouse double-click and enter 80 . Press <Enter>.
 - 3.5. Enter the parameter change interval in the **Interval parameter** field
 $t = [-50;50]$. Press <Enter>.
4. Specify function for the Z coordinate.
 - 4.1. Choose the **Constant** function type in the **Function by Z** list.
 - 4.2. Enter value **0** in the *Value by Z* field.
5. Click the **Create Object** button on the Advanced Control Panel.

Example 3

When designing a gear mechanism, you should determine the gear wheel's profile. The wheel's gear is drawn along the evolute of circle.

Wheel's parameters:

- ▼ wheel's module $m = 3$,
- ▼ number of the wheel's teeth $z = 20$,
- ▼ profile's angle $\alpha = 20^\circ$.

Parametric representation of evolvent in the cylindrical coordinate system:

$$R = D_b / \cos(t) / 2 \text{ (radius of the pitch circle),}$$

$$A = \operatorname{tg}(t) - t \text{ (involute angle),}$$

$$Z = 0,$$

where $0 < t < 0,4$ (in radians).

We recommend to do the following to create an involute.

1. Create variables in the Variables Window (see Table 2.5.11).

Tab. 2.5.11. Variables for involute construction

Name	Expression	Comment
m	3	wheel's module
z	20	number of teeth
a	20	profile's angle
D	$m * z$	diameter of the pitch circle
Db	$\operatorname{cosd}(a) * D$	diameter of the main circle



2. Call the **Curve by** command.



3. Select a coordinate type of the curve. To do this, activate the **Cylindrical Coordinates (R, A, Z)** switch in the **Coordinate Type** group on the Properties bar.

4. Specify function for the R coordinate.

- 4.1. Choose the **By expression** function type in the **Function by R** list. Two fields appear on the Properties bar: the field for entering function's expression and the field for entering the function's parameter name and its change interval.
- 4.2. Enter the function $\mathbf{Db / \cos(t) / 2}$ expression in the *Function expression by R* field.
- 4.3. Press <Enter>. The **Values of the parameters** table will appear on the Property Bar. Expression and Db variable value are transferred to the table from the Variables Window.
- 4.4. Enter the parameter change interval in the **Interval parameter** field $t = [0;0.4]$. Press <Enter>.

5. Specify function for the A coordinate.

- 5.1. Choose the **By expression** function type in the **Function by A** list.
- 5.2. Enter the function $\mathbf{\tan(t)-t}$ expression in the *Function expression by A* field. The t parameter change interval is defined automatically and is the same as the previously set interval for R coordinate.
- 5.3. Press <Enter>.

6. Specify function for the Z coordinate.
 - 6.1. Choose the **Constant** function type in the **Function by Z** list.
 - 6.2. Enter value **0** in the *Value by Z* field.
7. Click the **Create Object** button on the Advanced Control Panel.



Entering variables in the Variables windows is generally not obligatory. In this example, it is done to simplify expression for R coordinate.

2.5.4.13. Curve Rounding

You can round an angle formed by two curves. Depending on the relative position of the curves, the fillet is made by a circular arc of specific radius or by a spline lying on a cylindrical surface of specific radius. 3D lines, edges of bodies and surfaces, and sketch lines may be used as source curves.



It is impossible to round parallel segments, as well as concentric arcs.



To round two curves, apply the **Round Curves** command.

The elements for managing curve rounding will appear on the Properties Bar (Fig. 2.5.40).

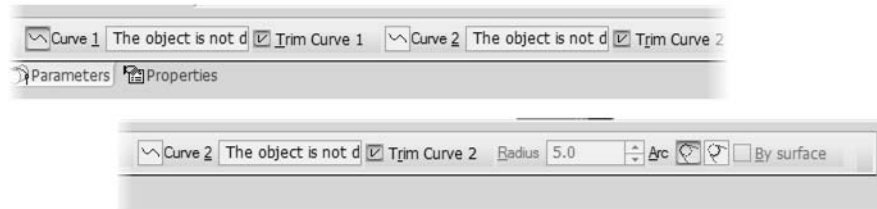


Fig. 2.5.40. Property Bar for rounding two curves

Specify the source curves in the model window. The names of selected objects will be displayed in the **Curve 1** and **Curve 2** fields.



Source curves should be specified in points located close to the assumed mating point.

After selection of source curves, a ghost of rounding i. e. a circular arc with characteristic point or a cylinder with a characteristic point and a spline will appear on the screen.

The rounding curve is constructed in such a way that its points of tangency with source curves are close to the selection points. Radius of curve rounding is defined automatically and its value is displayed in the **Radius** field. You may define the required fillet radius in the following ways:

- ▼ to enter the radius value in the **Radius** field or to specify it by the counter;
- ▼ to drag the defining point with the mouse.

Radius changing shall be displayed on the fillet ghost in the model window.

By default, trimming of source curves is enabled. In this case a copy is created for the source curve. The curve copy is trimmed in the point of its tangency with the fillet curve. You may disable the trim. Only a fillet curve shall be created in this case.



If the curve trimming option offered by the system differs from the expected one, disable trimming and use the **Curve trimming** command to delete parts of the curves (see Section 2.5.4.16 on p. 414).

The switches of the **Arc** group serve to select a desired direction of rounding.

If edges of one face is selected as the source curves, then you can place their fillet on the same face. To do this, enable the **By Surface** option.

Rounding of other curves on the surface can only be placed on it if the curves are **direct derivatives** with respect to the object containing the surface (see Section 2.1.2.7.2 on p. 114 for more information about viewing objects).



To change source curves, click **Select Other** button on the Special Control Panel and re-select objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the fillet, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361).

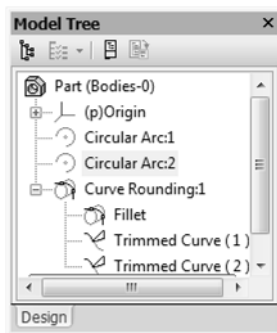


To finalize the construction, click the **Create Object** button on the Special Control Panel.

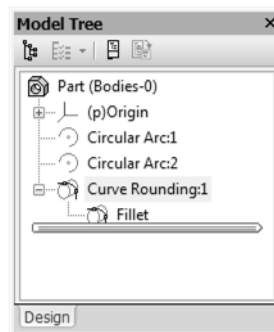


The mating curve appears in the model window and the curve's icon appears in the Design Tree.

- ▼ If source curves were rounded with trimming, then the Design Tree's branch corresponding to this curve rounding will contain subordinated objects: rounding curve and trimmed curves (Fig. 2.5.41, a).
- ▼ If the source curves were rounded without trimming, then the branch of the Design Tree corresponding to this curve rounding will have only one slave object: the rounding curve (Fig. 2.5.41, b).



a)



b)

Fig. 2.5.41. Display of rounding of a pair of curves in the Design Tree:
a) rounding with trimming; b) rounding without trimming



If no fillet ghost is displayed after specifying the source curves then it means that the selected curves can not be filleted close to the specified points. You should change the fillet radius. If the fillet ghost does not appear on the screen after changing the radius, then you should re-define source curves in other points.

2.5.4.14. Joining Curves

You can join vertices of two curves or start and end vertices of a single curve by a joining curve. The following may be used as source curves:

- ▼ 3D lines;
- ▼ edges of bodies and surfaces;
- ▼ contours in a sketch.

You can specify the following mating conditions in the point of joining of a source curve and a joining curve: **Not Specified**, **By Tangent**, **Perpendicularly** and **Smooth** (see Section 2.5.1.7 on p. 332).

You can change the form of a joining curve by means of tension parameter.



To join two curves, call the **Join Curves** command.

Control elements for joining curves will appear on the Properties Bar (Fig. 2.5.42).

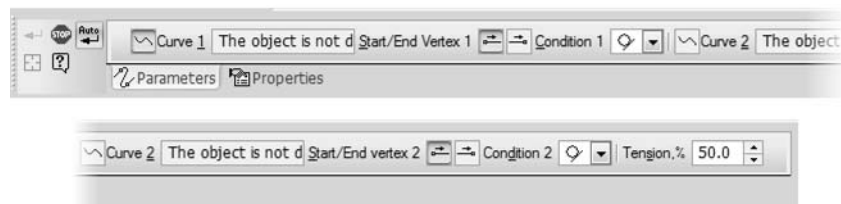


Fig. 2.5.42. Property Bar for joining two curves

Specify the source curves in the model window.



A initial source should be specified in a point that is close to the vertex where the joining to the other curve is assumed to be made.

If a 3D line is used as a source curve then it may be specified in the Design Tree. In this case the start point of the curve shall be automatically selected as the joining point.



Source curves may be also selected before the command selection.

To connect the ends of a single curve, you should specify it twice in the model window: near the start vertex and near the end vertex.

By default, the Object Auto Create option is enabled (see Section 1.4.2.10 on p. 93). In this case, the vertices of a curve (or curves) are joined immediately after they selected. Joining will be made with default settings:

- ▼ condition of mating a source curve and a joining curve in both joining points: by tangent;
- ▼ joining curve tension value is 50%.



The joining curve appears in the Model window and the curve icon will appear in the Design Tree. Mate condition icons to appear near the joining curve vertices.



You may control parameters of the joining curve during its creation. To do this, after calling the **Join Curves** command, disable the object auto creation mode (release the **Object Auto Create** button on the Special Control Panel). Then specify the source curves. A ghost of the joining curve with default parameters will appear on the screen.

Select the necessary mating condition for the joining and source curves. To do this expand the **Condition 1** (or **Condition 2**) list on the Properties panel and specify the necessary row.

Set the necessary shape of the joining curve (see Section 2.5.4.14.1 on p. 412).

If needed, you can change the mating vertex of the source curve (see Section 2.5.4.14.2 on p. 412).

All changes are displayed in a ghost of the joining curve in the model window.



To change source curves, click **Select Other** button on the Special Control Panel and re-select objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the joining curve, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361).



To finish the construction of the joining curve click the **Create Object** button on the Special Control Panel.

2.5.4.14.1. Specifying Tension for Joining Curve

You can change the form of a joining curve by means of tension parameter. The value of this setting is defined in percentage from 0 to 100.

- ▼ The less the tension setting, the less the curvature of the joining curve in the joining point: the curve is more shallow near the joining points.
- ▼ The more the tension setting, the more the curvature of the joining curve in the joining point: the curve is more shallow in its middle part.

Enter the desired tension value in the **Tension%** field on the Property Bar or specify it by a counter.

2.5.4.14.2. Changing Mating Vertex of Source Curve



If needed, you can change the mating vertex of a source curve. To do this, use the **Start/End Vertex 1** group of switches (or the **Start/End Vertex 2** group of switches). Enable a desired switch in the group corresponding to a source curve.



2.5.4.15. Curve by two projections

You can create curve based on two sketches situated in non-parallel flats. Sketch lines are considered as orthogonal projections of the curve to be created on the sketches' plane. Therefore, the curve is created as a two surface intersection curve; the surfaces are formed by moving specified projections in directions perpendicular to their planes (see Fig. 2.5.43

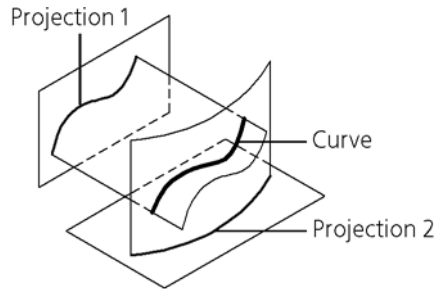


Fig. 2.5.43. Construction of a curve by two projections



To construct the curve by its two projections, call the **Curve by Two Projections** command. Select the first projection in the model window—a sketch or separate lines in it.



Specify the sketch in the Design Tree or Model Window. In the latter case it is necessary to preliminarily enable the sketch filter using the **Filter Sketches** context menu command (for more information on filtering by object, see Section 2.1.4.1.3 on p. 133). Note that filter is active during the curve by two projections creation only.



First projection lines can be selected also before you call the command.

The name of the object or the number of objects are shown in the **Projection 1** field on the Property Bar.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.

Select the second projection—a sketch or separate lines in it. The name of the object or the number of objects are shown in the **Projection 2** field.

The curve ghost will be displayed.



The absence of ghost is an indication that no curve can exist with the selected source objects.

To cancel object selection, re-select it.

All the changes will be reflected in the curve ghost.

The created curve may consist of one or several contours. Select the contours required for construction (see Section 2.5.4.15.1 on p. 414).



To change the projection objects click the **Select Other** button on the Advanced Control panel and re-select the objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the curve, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361).



To finish the construction of the curve by two projections, click the **Create Object** button in the Special Control Panel.

The created curve by two projections will appear in the model window, and intersection icon appears in the Design Tree.

2.5.4.15.1. Selection of contours for curve creation

The created curve may consist of one or several contours.

The **All Contours** option on the Property Bar enables selection of the necessary contours for construction. By default, the option is enabled. This means that all the resulting contours will be constructed.

If you need to keep only some of the contours disable the **All Contours** option. The **Contours List** panel containing a list of contours will appear on the Properties bar. Available options are displayed to the left from the contour names. By default all the options are enabled. Selecting a corner from the list will highlight it in the Model window. Disable the options of the contours which are not to be created.



To use or exclude most of the contours operate buttons **Enable All** and **Disable All** above the list.

2.5.4.16. Curve Trimming

You may construct a trimmed copy of a curve.

Open curve, 3D curve, contour in a sketch, edge of a surface or a body 3D curve (or its segment), sketch line, edge.

The following may be used as the secant object:

- ▼ any point object lying on the curve being trimmed (origin point, point in space, vertices of lines in sketches, separate points in sketches, vertices of 3D lines, vertices of edges etc.);
- ▼ object intersecting the source curve:
 - ▼ coordinate or auxiliary plane;
 - ▼ coordinate or auxiliary axis;
 - ▼ 3D line;
 - ▼ edge;
 - ▼ face;
 - ▼ a sketch line.

The source curve and the secant object should have a common point not coinciding with any vertex of the source curve.



To create a trimmed curve, call the **Trim Curve** command.

The elements for managing curve trimming will appear on the Properties Bar (Fig. 2.5.44).

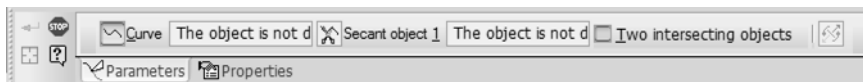


Fig. 2.5.44. Property Bar for curve trimming

Specify the source curve in the model window. You can select a 3D curve in the Design Tree.



A source curve may be also selected before the command selection.

Specify the secant object in the model window or in the Design Tree.

After selection of the source curve and the secant object a copy of the source curve shall be created that is split in two parts by the intersecting point. The part of the copy that does not contain the selection point of the source curve shall be displayed as a ghost on the screen. The ghost shall show the result of the source curve trimming. It is limited by two points: the intersection point and the end point of the source curve (Fig. 2.5.45).

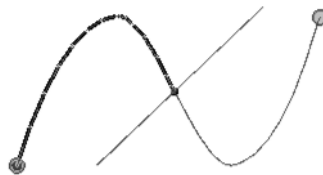


Fig. 2.5.45. A ghost of trimmed curve



Tangential point of the copy of source curve and the secant object is considered to be the intersecting point.



The **Change Direction** switch allows to select another part of the curve for the deletion, i. e. the opposite one in relation to the intersecting point.



You may also change the direction of trimming by specifying the other end point of the source curve in the model window.

If the source curve and the secant object have several common points then a ghost of the trimmed curve and the intersection point of selected objects shall be displayed. In this case the ghost of the trimmed curve is limited by the first intersection point **1** and the end point of source curve **2**. You may select other points **1** and **2** that limit the ghost.

- ▼ To specify a new position of point **1**, specify the required intersection point in the model window (Fig. 2.5.46).

- ▼ To set a new position of point **2**, specify the required intersection point holding the **<Ctrl>** key

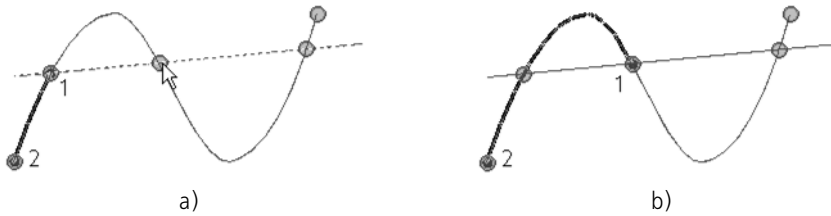


Fig. 2.5.46. Changing of point **1** position:
a) selection of intersection point; b) result

pressed (Fig. 2.5.47).

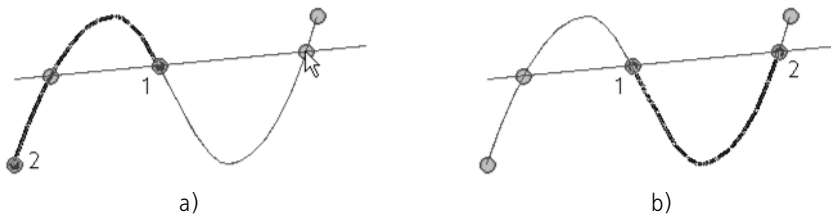


Fig. 2.5.47. Changing of point **2** position:
a) selection of intersection point; b) result



To change the source curve and the secant object, click **Select Other** button on the Special Control Panel and re-select objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the trimmed curve, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361).



To finish the construction of the trimmed curve click the **Create Object** button on the Special Control Panel.



The trimmed copy of the source curve will appear in the Model window and the icon of trimming will appear in the Design Tree.



A 3D curve is automatically hidden after trimming.

You can trim a source curve by two secant objects (see Section 2.5.4.16.1 on p. 416).

2.5.4.16.1. Curve Trimming by Two Objects

You can create a copy of the curve trimmed by two objects. To do this, enable the **Two Intersecting Objects** option. The **Secant Object 2** switch will appear on the Property Bar, the **Change Direction** switch will disappear (Fig. 2.5.48).

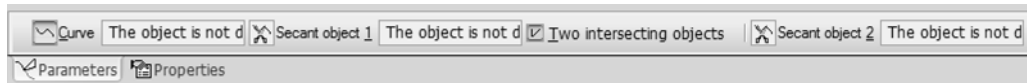


Fig. 2.5.48. Property Bar for curve trimming by two objects

Select the source curve and the secant objects in the model window or the Design Tree. A trimmed curve ghost limited by points of intersection of the source curve and the secant objects **1** and **2** will appear on the screen (Fig. 2.5.49).

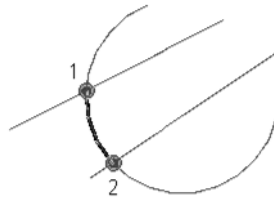


Fig. 2.5.49. Trimmed curve ghost limited by points of intersection of source curve with secant objects

If the second secant object intersects the source curve in several points, then only the first intersection point will be displayed on the screen, other ones will be hidden (Fig. 2.5.50,a). To display them, press the <Ctrl> key. In this case all points of intersection with the first secant object will disappear, except for the point limiting the ghost (Fig. 2.5.50, b).



Fig. 2.5.50. Ghost of curve trimmed by two objects:
a) without pressing the <Ctrl> button; b) with pressing the <Ctrl> button

You can change the position of points **1** and **2** by specifying the desired intersection points.



Note that the ghost of a trimmed curve is always limited by points of different secant objects.

2.5.4.17. Equidistant Line

You may build a curve which is equidistant to the base one.

The following objects may be used as a base curve:

- ▼ a 3D curve,
- ▼ an edge,

- ▼ a sketch line,
- ▼ Chain of objects listed above in any combination.



To create an equidistant curve run the **Equidistant Line**.
Select the base curve in the model window.



A base curve may be also selected before the command selection.

A ghost of the equidistant curve with default parameters will appear on the screen. The base curve start vertex is marked with an arrow.

- ▼ When you select the curve for the second, it becomes deselected.
- ▼ The second selection of a curve from the set makes deselected this curve together with all other curves towards the end vertex.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.



The name or number of selected objects is displayed in a field beside the **Curve** switch.



An equidistant curve can be constructed in two ways:

- ▼ **Offset Away from Vertex** (see Section 2.5.4.17.1);
- ▼ **Offset Along Surface** (see Section 2.5.4.17.2).



Select the equidistant line construction method from the **Construction Method** drop-down list on the Properties bar.

Then specify the objects and parameters for constructing an equidistant curve.

All changes are displayed in the curve ghost in the model window.



To change the objects used for constructing the equidistant curve by the selected method click the **Select Other** button on the Advanced Control panel and re-select objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the curve, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361).



To finalize the construction, click the **Create Object** button on the Special Control Panel.



The equidistant curve appears in the Model window and the curve icon appears in the Design Tree.



At zero offset the equidistant surface copies the base surface.
Such copy remains integral even if a set of curves has been selected as a base curve.

2.5.4.17.1. Offset Away from Vertex

This method enables setting the position of the equidistant curve by setting off its initial (final) vertex relative to the corresponding vertex of the base curve. The offset operation is performed in the offset plane following the offset vector.

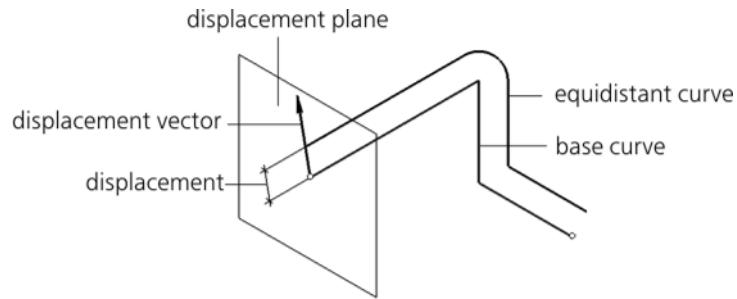


Fig. 2.5.51. Equidistant curve creation by **Offset from the Vertex method**;

Offset plane— is a plane perpendicular to the base plane in its start or end vertex.

The direction of a **offset vector** is specified by means of guide objects.

Guide Objects can be represented by the following objects in the model.

▼ Point objects:

The direction defined by a point object is projection of a line, passing through this point object and the start/end vertex of the base curve, on the offset plane.

▼ Straight Objects.

The direction defined by a straight object is an object projection on the offset plane.

▼ Flat objects.

The direction defined by a plane object is projection of a straight line on the offset object.

Point, straight and flat objects are listed in Table 2.1.7 on p. 131.

In addition, the guiding object may be represented by a vector (see Section 2.8.5 on p. 613).

To create an equidistant line, complete the following steps.

1. Specify the base curve (see Section 2.5.4.17).

Then, the guide object is defined automatically — one of the coordinate axes of the current coordinate system. The name of the axis will display in the field next to the **Direction** switch on the Properties bar.

If necessary, you can select some other guiding object. To do so, activate the **Direction** switch and select the desired guide object in the Design Tree or in the Model window, or build a vector.

The name of the selected object will be displayed in the field next to the **Direction** switch. If the direction is set by means of a vector, the field will contain the word “Vector”.

The set direction is considered default — i.e. initial.

The equidistant line's ghost appears in the model. Its vertex will be at the default distance from the base curve in the initial direction.

2. Set parameters of the offset using any method.

▼ **Manual setting**

In the **Angle** field of the Property bar, enter the rotation angle's value for the offset vector in the offset plane. The angle is calculated from the initial direction.



In the **Distance** field, enter the distance between the base curve and the one being created in the offset plane.



The angle value can be set using a characteristic point (see Section 2.1.4.2.2).

▼ By specifying a point



Click the **Object** button in the Property bar. Then, specify the point object.

The values defining position of the starting (ending) point of the equidistant line are displayed in the **Angle** and **Distance** field. The angle is calculated from the initial direction.

- ▼ If the **Associate** option is enabled, specifying the point object results in forming of the associative link between the starting (ending) point of the equidistant line with that object.



The presence of relation is manifested by displaying a checkmark on the **Object** button and the name of the point object in the **Object** field on the Property Bar. The **Angle** and **Distance** field will not be available for editing in this case.

Thanks to the associative link, the newly created equidistant line will be changing its form while the point is being moved in space.

To cancel the associative link to the point object, click the **Object** button. The tick on the button will disappear.

- ▼ If this option is disabled, the equidistant line without the link to object will be created when specifying the point object. There will be no checkmark on the **Object** button, the object name will not be displayed, and **Angle** and **Distance** field will not be available for editing.



To restore the associative link between the equidistant line and a point object, enable the **Associate** option, click the **Object** button, and reselect the required object.

3. Set the remaining parameters.



- ▼ The **Corner passing** group of switches makes it possible to choose the method of joining the equidistant curve's segments (see Fig. 2.5.52).



Fig. 2.5.52. Corner Treatment Methods:
a) passing with fillet; b) passing with cut



- ▼ The **Retain radii** button controls creation of the equidistant line by segments of the base curve, mated with circles' curves (for example, radii in a polyline vertices).

If the button is disabled, the equidistant line will be built in these segments with radii that will differ from the radii of the base curve. Their values are calculated automatically (Fig. 2.5.53, a). If the button is enabled, the equidistant line's radii will be the same as the radii of the respective arc segments of the base curve (Fig. 2.5.53, b).



Fig. 2.5.53. Rounding segments of an equidistant curve: a) without saving radii; b) with saving radii



- ▼ The **Initial/Final Vertex** group of switches enables changing the base curve vertex through which the offset plane passes.



2.5.4.17.2. Constructing by Offset Along Curve

This method enables setting the position of the equidistant curve by its offset along the theoretical surface of the specified face to the specified distance from the base curve (theoretical surface is detailed in Section 2.5.1.3 on p. 328). The base curve must belong to the specified face.



It is recommended to specify the curve being the initial or derived object of the specified surface as the base curve.

Example on the use of source object: a curve constructed equidistantly to the curve used for constructing a surface using a curve grid (see Section 2.6.1.8 on p. 458) or a ruled surface (see Section 2.6.1.9 on p. 461).

Example on the use of derivative object: a curve constructed equidistantly to the surface intersection curve.



The distance between the equidistant curve and the base one is measured in the direct line and not along the theoretical surface of the face (Fig. 2.5.54).

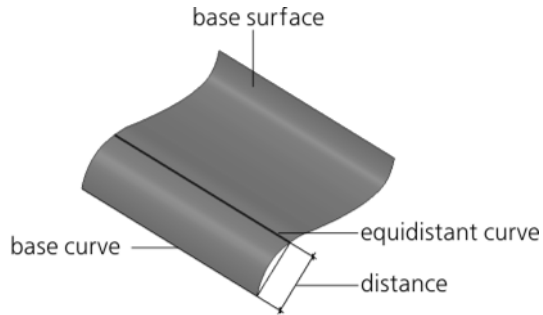


Fig. 2.5.54. Equidistant curve constructed using the **Offset Along Surface** method

Specify the face for constructing an equidistant curve.



The specified face is highlighted in the Model window and its name is displayed in the field adjacent to the **Surface** switch.

Set the distance from the base curve to the one being created by entering it into the **Distance** field.



The distance value can be set using a handle (see Section 2.1.4.2.2).

2.5.4.18. Isoparametric Curve

Allows to construct isoparametric curve of a theoretical surface of the specified face going through the selected point of this face.

For more information on theoretical face surface and isoparametric curves see Section 2.5.1.3 on p. 328.

Point position on a face can be specified using two methods:

- ▼ by specifying parameters U and V of the point of the theoretical face surface,
- ▼ by specifying a point object on the specified face within its boundaries; as a point object, you can use:
 - ▼ object's vertex (or a point in space) provided that the object (or point) is **the direct source or direct derivative** with respect to the object, containing the face¹.
 - ▼ face edge vertex.

To create an isoparametric curve, complete the following steps.



1. Call the **Isoparametric Curve** command.
2. Select the desired face in the Model window.



Base face for constructing an isoparametric curve can be also selected before calling the command.

1. It means that in the hierarchy of relationships between the object containing the vertex (or point) and the object containing the face, no more objects should be present (see Section 2.1.2.7.2 on p. 114 for more information about viewing objects' relationships).

The selected face is highlighted. The following will appear on the selected face: ghost of the face indication point and ghosts of isoparametric curves U and V going through this point. The **Surface** field of the Property Bar displays the name of the selected face. The input fields **U Parameter, %** and **V Parameter, %** display values of the U and V parameters at the face indication point. The value counting direction is shown by arrows from the start point U=0, V=0.

- Specify the position of the point through which the isoparametric curve must pass. This can be completed by the following methods:

Method 1

Enter the required values of the U and V parameters in percent (from 0 to 100) in the **U Parameter, %** and **V Parameter, %** fields or use the counter to specify them. All updates shall be displayed on the LCS ghost in the model window. The location of the point also can be changed by moving its handle with the mouse (see Section 2.1.4.2.2).

Method 2

Select the required point object in the Design Tree or in the model window.



The **Object** button will be ticked, and the field next to the button will display the object name. This indicates the associative relation of the point, through which a straight line should pass, with the selected object. Due to this relation, the created curve will follow the object in case it changes its position. If necessary, the relation can be disabled by clicking the **Object** button. The tick on the button will disappear, and the field will be cleared.



The object selection is detailed in Section 2.1.4.1.1 on p. 130.

- Select one of two curves for creation.



By default, isoparametric curve of direction U is created— its ghost is active. In this case, the **Curve of Direction U** switch is active in the **Select by Direction** group of switches of the Properties bar.



To create an isoparametric curve of direction V, activate the **Curve of Direction V** switch. The ghost of curve of direction V becomes active.

- If required, you can also:



- Cancel all objects set for construction of a curve. To do this, click the **Reselect** button on the Advanced Control panel.

- Change name and color of the created curve. Enable the **Properties** tab of the Property Bar. Display parameters of the curve can also be set in the same tab (display parameters are detailed in Section 2.5.4.1 on p. 361).



- Click the **Create Object** button on the Special Control Panel.

The isoparametric curve appears in the model window, while its icon appears in the Design Tree. After creating the first curve, the system waits until the user specifies the objects for the next curve.

2.5.4.19. Group of isoparametric curves on the surface

You can create a group of isoparametric curves on the theoretical surface of the selected face. Number of curves of each direction (U and V) in a group are defined by the user and may be different.

For more information on theoretical face surface and isoparametric curves see Section 2.5.1.3 on p. 328.



To create a group of isoparametric curves, call the **Group of Isoparametric Curves** on the Surface command.

Select the desired face in the Model window.



Base face for constructing isoparametric curves can be also selected before calling the command.

The selected face is highlighted. Ghosts of isoparametric curves of U and V directions appear on it. Arrows show the directions of U and V parameters effect in the start point $U=0, V=0$; The **Surface** field of the Property Bar displays the name of the selected face.

Enter the desired number of curves of each direction into the **Number, U** and **Number, V** fields of the Properties bar.

The number of curves of surface parameter direction U (V) is entered in the **Number, U** (**Number, V**) field. It is the curves of direction V(U) positioned with equal spacing by parameter U(V).



The **Boundaries** group of switches controls construction of boundary curves corresponding to the boundaries of theoretical surface. **Save Boundaries** option button is active in this group by default. It means, the group of curves will be created with edged curves.



If you need to create a group without edged curves, activate the **Delete Boundaries** switch.

The **Explode** option controls the result of curve group construction.

- ▼ If option is disabled, after completion of the command curves will be displayed in the Design Tree as objects subordinated to the group.
- ▼ If the option is enabled, after completion of the command, curves will not be subordinated to the group, and each curve will be displayed in the Design Tree as a standalone object.



To change the face used for constructing a group of curves click the **Select Other** button on the Advanced Control panel and re-select the face.

You can change the name, color and display parameters of curves of group in the **Properties** tab of the Properties bar (display parameters are detailed in Section 2.5.4.1 on p. 361).



To finish the construction of the group of isoparametric curves, click the **Create Object** button in the Special Control Panel.

The isoparametric curves appear in the model window, the group of isoparametric curves or the set of separate isoparametric curves in the Design Tree (if the **Explode** option was enabled).

To expand the list of curves of the group, click the mouse on the “plus” sign located at the left of the group icon in the Design Tree.

At first, the Design Tree contains V-direction isoparametric curves splitting the face in U direction and then U-direction isoparametric curves splitting the face in V direction.



Please note that option **Explode** is available on the Property bar only when creating the group. If you need to transform already created group into a set of individual curves, you may explode the group. Group breakup is performed similarly to breaking up an array (see Section 2.7.1.3.7 on p. 517). After group breakup, every curve is displayed in the Design Tree as an individual object – isoparametric curve.

2.5.4.20. Outline Line

You can build the **face outline line** which is the line in whose points the face's normal is perpendicular to the specified sight direction.

Face edges are not included into the outline line.



You cannot build the online line of a face, if the face is flat or the face's normal in each its point is perpendicular to the direction of sight.

Outline line can be created for a set of faces (see Fig. 2.5.55).

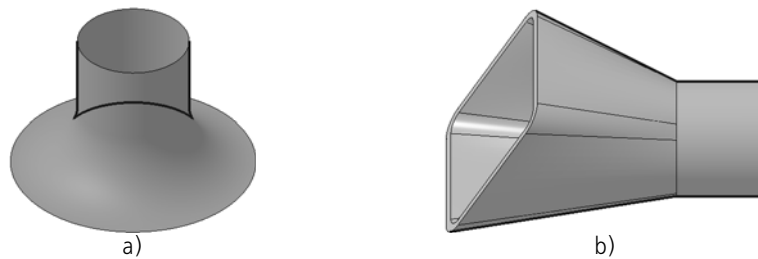


Fig. 2.5.55. Outline Line: a) of one face; b) of set of faces

To create a face outline line perform the following operations.



1. Call the **Outline Line** command
2. After calling the command, the **Surface** switch is enabled on the Property bar automatically. Select the required face(s) in the Model window.

If you need to create an outline line for all faces of a body or a surface, select this body or surface in the Design Tree.



Source objects may be also selected before the command selection.

The selected objects are highlighted. The name of the selected object or the number of selected objects are shown in the **Surface** field on the Property Bar.

The second selection of a face cancels it's selection.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.

The outline line ghost will be displayed. By default, an outline line is constructed in the view direction perpendicular to the screen plane.

3. If you need to fix the outline line in the form corresponding to the model current position, enable the **Fix Position** option. With the option disabled the outline line will be reconstructed each time after changing the model orientation.

4. Select the guide object or create a vector in order to specify the view direction (see Section 2.5.4.20.1 on p. 426).

After specifying view direction, the ghost of the outline line changes.

5. An outline line may consist of one or several segments— contours. Select the contours required for outline line construction (see Section 2.5.4.15.1 on p. 414).

6. If required, you can also:



- ▼ Cancel all objects set for construction of a curve. To do this, click the **Reselect** button on the Advanced Control panel.

- ▼ Change name and color of the created curve. Enable the **Properties** tab of the Property Bar. Display parameters of the curve can also be set in the same tab (display parameters are detailed in Section 2.5.4.1 on p. 361).



7. Click the **Create Object** button on the Special Control Panel.

The created outline line will appear in the model window, and its icon will appear in the Design Tree.

2.5.4.20.1. Setup of view direction when creating an outline line.

By default, an outline line is constructed in the view direction perpendicular to the screen plane. You can specify view direction using a guide object or a vector.

Any straight or flat object may function as a guide object. Straight and flat objects are listed in Table 2.1.7 on p. 131.

Direction, defined by a straight object — a line parallel to the object.

Direction, defined by a plane object — a line perpendicular to the object.



Activate the **View Direction** switch and then select an object defining the view direction in the Design Tree or in the model window.



Reselecting of the object cancels its selection and the view direction becomes default— perpendicular to the screen plane.



To create a view vector, click the **Vector drawing** on the Advanced Control Panel (see Section 2.8.5 on p. 613).

The selected object's name will be displayed in the **Projection Direction** field. If the direction is set by means of a vector, the field will contain the word "Vector".

2.5.4.21. Spline on Surface

You can create a spline on the theoretical surface of the selected face. If necessary, face theoretical surface is extended.

For more information on theoretical face surface, see Section 2.5.1.3 on p. 328.

The spline vertex can be constructed as follows:

- ▼ by specifying parameters U and V;
- ▼ by specifying a point object on the specified face within its boundaries; as a point object, you can use:
 - ▼ object's vertex (or a point in space) provided that the object (or point) is **the direct source** or **direct derivative** with respect to the object, containing the face¹.
 - ▼ face edge vertex.

To construct a spline on surface, perform the following steps.



1. Call the **Spline on Surface** command.
2. Specify a face on which it will be located in the model window.



Base face for constructing a spline can be also selected before calling the command.

The selected face is highlighted. Name of the selected object shall appear in the **Surface** field on Properties toolbar.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.

3. Set spline vertices. This can be completed by the following methods.

Method 1

Specify the vertex position on the face by a mouse click or enter the required values of the U and V parameters in percent (from 0 to 100) in the **U Parameter, %** and **V Parameter, %** fields.

Method 2

Select the required point object in the Design Tree or in the model window.



The **Object** button will be ticked, and the field next to the button will display the object name. This indicates the associative relation of the spline vertex with the selected object. Due to this relation, the created curve will follow the object in case it changes its position. If necessary, the relation can be disabled by clicking the **Object** button. The tick on the button will disappear, and the field will be cleared.



The object selection is detailed in Section 2.1.4.1.1 on p. 130.

Each vertex of the spline is characterized by a set of parameters displayed in the Table of vertex parameters. You can edit the parameters of existing vertices and create new vertices using table controls and table context menu commands (see Section 2.5.4.21.1 on p. 428).



The spline with the current parameters is displayed on the screen as a ghost.



4. Select an option of curve construction— closed or open. To do this, activate the desired switch in the **Mode** group: **Open Curve** or **Closed Curve**.



5. Select the spline type. To do this, activate the desired switch in the **Type** group: **Spline by Vertices** or **Spline by Poles** (for details about spline types, see Section 2.5.4.9.1 on p. 387).



6. If required, you can also:

1. It means that in the hierarchy of relationships between the object containing the vertex (or point) and the object containing the face, no more objects should be present (see Section 2.1.2.7.2 on p. 114 for more information about viewing objects' relationships).

- ▼ Change of position for the preset vertices and control of the spline by vertices shape. To do this, select a row with the number of desired vertice in the parameters table. The system is switching to a spline editing mode (for more about spline editing see Section 2.5.4.21.2 on p. 429). You can delete spline vertices in this mode. To activate the spline construction mode, select the last (unnumbered) row of the table. In addition, you can switch between the construction and editing spline modes by using the **Finish/Continue Vertex Input** context menu command.



- ▼ Change of face for constructing a curve To do this, click the **Reselect** button of the Special Control Panel and select other face and spline vertices.

- ▼ Change name and color of the created curve. Enable the **Properties** tab of the Property Bar. Display parameters of the curve can also be set in the same tab (display parameters are detailed in Section 2.5.4.1 on p. 361).



7. Click the **Create Object** button on the Special Control Panel.

The created spline will appear in the model window whereas its icon will be displayed in the Design Tree.

2.5.4.21.1. Table of Spline Vertex Parameters

Every spline vertex is characterized by a set of parameters. Their values are displayed in the Table of Vertices.

This table is called **Vertices Coordinates (U,V)** and is located on the Properties Bar.

Each spline vertex corresponds to a table row.

The table contains the following parameters:

- ▼ Vertex number,
- ▼ the values of vertex parameters (U,V),
- ▼ Relation mark is the spline vertex is associated with a point object,
- ▼ weight (while constructing the spline by poles);
- ▼ relation and system pole marks (while constructing the spline by poles).

System pole of a spline on surface is similar to a system pole of a spline mated **By Tangent** with a model object. For more information on the system pole, see Section 2.5.4.9.3 on p. 389.

N		U	V	Weight
1	<input type="checkbox"/>	36.0	23.0	1.0
2	<input checked="" type="checkbox"/>	100.0	0.0	1.0
3	<input checked="" type="checkbox"/>	75.0	20.0	1.0
4	<input checked="" type="checkbox"/>	81.8514	100.0	1.0
	<input type="checkbox"/>			

Apex coordinates (UV) << Management

Fig. 2.5.56. Sample Spline Vertex Parameter Table

The table rows are formed automatically during vertex construction in the model window. At the same time, creating or editing the table row shall involve creation of a new table or will change the parameters for the current curve vertex. In addition, selection of row of existing vertex enables spline edit mode (for more about editing see Section 2.5.4.21.2 on p. 429). Selection of the last (unnumbered) row of the table enables spline creation mode.

Table controls and table context menu commands allow the following manipulations:




- ▼ replacing the entries in all cells of the selected column with the zero value or the first cell value;
- ▼ modify parameter values of existing vertices in the table,
- ▼ Create new vertices,
- ▼ Delete the curve's vertices,
- ▼ Allows deleting the connection of a spline vertex to a point object,
- ▼ cancel conditions of spline's tangency with the curve on the spline's surface (see Section 2.5.4.21.4 on p. 431 for more information on condition of the spline's tangency with the curve).

Creating of new spline vertex is detailed below, other methods of work are described in Section 2.5.4.8.2 on p. 377.

Creating a new vertex

To add new vertex, select a table row with the number of vertex in relation to which adding of vertex will be made, and click the button above the table. The description of buttons enabling to add a new vertex is given in Table 2.5.12.

Tab. 2.5.12. Buttons for spline vertices adding

The	Description
 Insert a vertex before the current	Allows to add a vertex between the current and previous. Upon clicking the button, new vertex appears and new row containing new vertex parameters is added to a table. New vertex gets the number of current vertex, numbers of following are increased by one. You may also add a new vertex by pressing the <i><Insert></i> key.
 Duplicate vertex	Allows to add a new vertex which coordinates and weight have to be the same as current vertex has (weight for spline by poles). Upon clicking the button, new vertex appears and new row containing this vertex parameters is added to a table. New vertex gets the number of current vertex, numbers of following are increased by one.
 Insert vertex after current one	Allows to add a new vertex between the current and next. Upon clicking the button, new vertex appears and new row containing new vertex parameters is added to a table. New vertex gets the number that follows in order the number of current vertex, numbers of following vertices are increased by one.

2.5.4.21.2. Spline edit mode

In editing mode you can:

- ▼ change the spline vertex position;

- ▼ control the form of the spline at its vertex – change direction and length of the tangent vector of the spline.

To switch to spline edit mode, select the row with desired vertex number in the table of vertices parameters.



The system automatically switches to the editing mode on running the edit command for an earlier created spline.

The **Control** group of switches on the Property bar allows to select the method of spline editing for the current vertex:



- ▼ **Vertex Position**—a change in vertex position (see Section 2.5.4.21.3 on p. 430);



- ▼ **Tangent vector on a vertex**—control of the spline's tangent vector on the current vertex (see Section 2.5.4.21.4 on p. 431).



In case of a spline By poles, you can control the tangency vector on extreme vertices only. Please note the following specifics of switching to a spline by poles edit mode. Highlighting the line of the initial or ending vertex allows to switch to editing the position of this vertex and to controlling its tangent vector— there is the **Control** group on the property bar. Selecting lines for other vertices makes it possible to switch to editing their positions— there are fields for entering U and V parameters on the property bar.

In addition, you can delete vertices in the edit mode. For this purpose, select the required vertex in the model window and press the *<Delete>* key.

All changes are displayed in the spline ghost in the model window.

After finishing editing spline at the one of vertices, you can:

- ▼ continue setting vertices— highlight the final (unnumbered) line of the table;
- ▼ edit other vertex— to do this, select its row in a table or specify it in a model window.
- ▼ finish the construction of the spline— to do this, click the **Create Object** button.



You can switch between the construction and editing spline modes by using the **Finish/Continue Vertex Input** context menu command.

2.5.4.21.3. Changing of vertex position



To change position of the current vertex of the spline, enable the **Vertex position** switch of the **Control** group on the Property bar.

Control elements for joining curves will appear on the Properties Bar (Fig. 2.5.57).

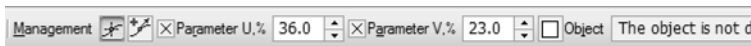


Fig. 2.5.57. Property bar when changing the position of the current vertex of the spline

There are two ways to change the position of the spline's vertex:

- ▼ specify parameters U and V;
- ▼ bind the vertex with the point object.

Methods of changing the spline's vertices' positions are similar to methods of setting a vertex when creating a spline. These methods are detailed in item 3 of Section 2.5.4.21 on p. 426.

2.5.4.21.4. Controlling the tangent vector of the spline

You can change the direction and length of the spline's tangent vector for the current vertex.

The spline's tangent vector for the current vertex can have arbitrary direction or tangent to the curve going through this vertex.

The following may be used as a curve:

- ▼ curve on selected face or edge of this face;
- ▼ isoparametric curve of a theoretical surface of the specified face.

You can change the length of the tangent vector as you wish.



Spline tangency condition setting

To switch to controlling the spline's tangent vector, enable the **Tangent vector in the vertex** switch in the **Control** group of the Property bar.

Control elements will appear on the property bar to make it possible to set conditions of spline tangency with respect to the current point. Select the required spline tangency condition from the **Tangency** list (Fig. 2.5.58).

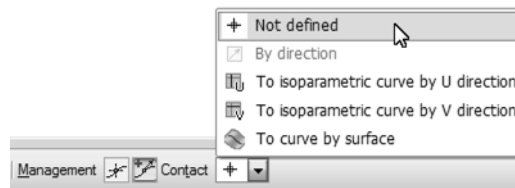







Fig. 2.5.58. Property bar when editing the tangent vector in the current vertex of the spline

Tangent vector control is described in Table. 2.5.13.

Tab. 2.5.13. Spline tangent vector control at the current vertex

	Tangency condition	Tangent vector control
	Not specified	After choosing this condition, spline shape at the current vertex takes a default view.
	By direction	Allows to manually edit parameters of the vector. Once the tangency option is selected, the tangency vector appears on the screen as a dashed segment with a handle. You can edit vector length and direction by moving with mouse its handle.

Tab. 2.5.13. Spline tangent vector control at the current vertex

	Tangency condition	Tangent vector control
	To isoparametric curve U;	Use it to set the vector's tangential direction with respect to U (or V) isoparametric curve of the specified surface. Once the above tangency condition is set, the vector will be positioned tangentially with respect to the isoparametric curve. Vector of the selected direction appears on the screen. You can edit vector length by moving with mouse its handle.
	To isoparametric curve V;	
	To a curve on surface	Use it to set the tangential direction of the vector with respect to a curve on the specified surface. After choosing this tangency condition, vector with default direction appears on the screen. Tangency designation will appear near such a vertex. Specify in the model window a curve or an edge which define vector direction. Then change vector length by moving with mouse its handle.

The ghost of the spline being created in the model window will be reconstructed according to the tangency condition selected and the tangency condition symbol will appear next to the vertex.



The **Change Direction** switch makes it possible to reverse the tangent vector direction.

Note that in case of a spline By poles, you can control the tangency vector on extreme vertices only. Once parameters of the tangent vector of the extreme vertex have been changed, the vertex adjacent to it is given the property of a system pole— a symbol of the system pole appears in the table of vertices' parameters in the line of this vertex (see Fig. 2.5.56 on p. 428). If vertices adjacent to the extreme ones are related to point objects, then after selection of the tangency condition, a message will appear on the screen indicating that parameters of these vertices have been changed.

For more information on the system pole, see Section 2.5.4.9.3 on p. 389.

Spline tangency condition deletion

You can delete spline tangency condition at the current vertex or at all the vertices simultaneously.



To delete a preset condition of the spline's tangency to the current vertex, select **Not set** from the **Tangency** list on the property bar or call the command of the contextual menu **Delete control parameters — In the point**.

To delete the specified tangency conditions at all the vertices use the **Delete Control Parameters — At All Points** drop-down menu command.

Please be advised that you may not restore the tangency condition once deleted. You may only redo the tangency condition in order to restore it.

If the spline vertex is assigned the system pole attribute, the tangency condition may be deleted by removing the attribute. To do so, highlight the system pole row in the vertices parameters table and click on the system pole icon in the **Link** column or use the **Delete Relation** com-

mand from the table drop-down menu. The tangency condition will be deleted. The vertex coordinates will remain unchanged.

2.5.4.22. Projection curve

You can create a **projection curve** – a projection of the selected curve on a projected surface.

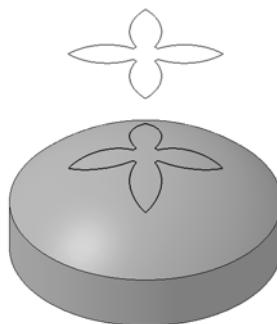


Fig. 2.5.59. Sketch projection on a body's face

2.5.4.22.1. Objects for construction

For construction of a projection curve, you will need a **projection surface** and a **base curve**.

The following can be used as the projection surface:

- ▼ face of the body or surface,
- ▼ Coordinate or auxiliary plane
- ▼ a set of bodies' faces and/or surfaces,

The following objects may be used as a base curve:

- ▼ a 3D curve (including array instances),
- ▼ body or surface edge,
- ▼ a sketch line,
- ▼ a set of the above-mentioned objects,
- ▼ coordinate or auxiliary axis,
- ▼ curve array,
- ▼ a group of isoparametric curves.



Please, pay attention to the following features of the projection line construction.

- ▼ It is impossible to create an axis projection on a plane.
- ▼ If a base curve projection on a plane degenerates into a point, it will be impossible to build it.
- ▼ Projection of an axis is possible, if this projection is located within the projection surface.

2.5.4.22.2. Construction of a projection curve

To create a projection curve, complete the following steps.



1. Call the **Projection Curve** command.



2. After calling the command, the **Surface** switch is enabled on the Property bar automatically. Select the required face or plane in the Model window. The object selection is detailed in Section 2.1.4.1.1 on p. 130.

The selected object is highlighted. Name of the selected object shall appear in the **Surface** field on Properties toolbar.



3. After specifying projection surface, the system will proceed to the selection of base curve, and the **Curve** switch will be activated automatically. Select the base curve in the model window. The name or number of objects is displayed in a field beside the switch.



Note that the axis, curve array, and the group of isoparametric curves may not be included in the set of objects making up the base curve. Therefore, selection of a new axis, for example, cancels selection of all other objects included in a base curve.

The curve ghost will be displayed.



The absence of ghost is an indication that no projection curve can exist with the selected source objects and current construction parameters.

4. If necessary, change the projection surface (for example, you can create a set of faces) and/or a base curve.



▼ In order to change the surface of projection, enable the **Surface** switch and then specify the required objects. A plane may not be included in the set of objects making up the surface of projecting. Therefore, selection of a new axis cancels selection of all other objects included in the surface of projecting.



If you need to add all faces of a surface or a body to the current face set, select the body or surface in the Design Tree.



▼ In order to change the base curve, enable the **Curve** switch and then specify the required objects.

To cancel object selection, re-select it.

All the changes will be reflected in the curve ghost.

5. Select the type of base curve projection: along the normal to selected surface or in the specified direction (see Section 2.5.4.22.3 on p. 435).

6. By default, projection of the reference curve is built within the contour of the face (trimmed by the faces contour) – **Trim by boundaries** option is enabled on the property bar. If you need to construct whole projection, disable the option. Base curve will be projected on the theoretical surface of the face. If necessary, theoretical surface of the face will be extended.

If a set of faces is used as the surface for projection, the base curve projection will be trimmed by the contour of each face of the set by default.



Please, pay attention to the following features of the projection.

- ▼ Base curve projection on a *surface* is always constructed in full.
- ▼ If an *axis* is specified as the base curve, its projection will always be constructed within the projected surface's contour.

In both cases, the **Trim by boundaries** option will not be available.

For more information on theoretical face surface, see Section 2.5.1.3 on p. 328. The Figure shows options for constructing sketch line (base curve) projection on the face with hole.



Fig. 2.5.60. Result of the sketch line projection on a face with a hole:
 a) the **Trim by borders** option is on;
 b) the **Trim by borders** option is off

7. A projection curve may consist of one or several segments - contours. Select the contours which need to be constructed (see Section 2.5.4.15.1 on p. 414).
8. If required, you can also:



- ▼ Cancel all objects set for construction of a curve. To do this, click the **Reselect** button on the Advanced Control panel.
- ▼ Change name and color of the created curve. Enable the **Properties** tab of the Property Bar. Display parameters of the curve can also be set in the same tab (display parameters are detailed in Section 2.5.4.1 on p. 361).



9. Click the **Create Object** button on the Special Control Panel.
 The projection curve appears in the model window, while its icon appears in the Design Tree.

2.5.4.22.3. Selection of the base curve projection type.



By default, the reference curve is projected by normal to the preset projection plane. In this case, the switch is active in the **Projection Type** group on the Property bar **Normal to Surface**.



You can create base curve projection in the direction specified by an object or a vector. To do this, activate the **By Direction** switch in the **Projection Type** group. The Property bar will display controls for setting up the guide object.

Any straight or flat object may function as a guide object. Straight and flat objects are listed in Table 2.1.7 on p. 131.

Direction, defined by a straight object — a line parallel to the object.

Direction, defined by a plane object — a line perpendicular to the object.



Activate the **Direction** switch and then select an object defining the direction of projection in the Design Tree or in the model window.

Reselecting of the object cancels its selection and the ghost of the curve being created disappears.



To create a vector specifying the direction of projection, click the **Vector drawing** on the Advanced Control Panel (see Section 2.8.5 on p. 613).

The selected object's name will be displayed in the **Projection Direction** field. If the direction is set by means of a vector, the field will contain the word "Vector".

The Figure shows an example of constructing sketch line (base curve) projection on the cylindrical surface.

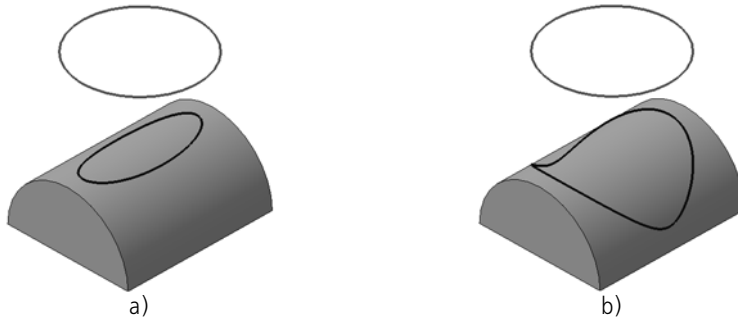


Fig. 2.5.61. Example of constructing sketch line projection on the cylindrical surface.
 a) projecting of a sketch line along the normal to surface;
 b) projecting of a sketch line perpendicularly to the sketch plane

2.5.4.23. Intersectional Curve of Surfaces

You can build an intersection curve for two surfaces, two sets of surfaces or for a surface and a set.



To build a surface intersection curve, run the **Surface Intersection Curve** command.



Once the command has been called, the **Set of surfaces 1** switch will be enabled automatically in the Property bar. Specify the first intersection object — a face or a surface (object selection is detailed in Section 2.1.4.1.1 on p. 130). The name of the selected object is displayed in the field beside the **Set of surfaces 1** switch.



After the first intersection object is specified, the system switches to the second intersection object and automatically activates the **Set of surfaces 2**. Specify the 2nd or several intersection object(s). The name or number of objects is displayed in a field beside the switch.

If you need to change the first or the second set of surfaces, enable the respective switch and specify the required objects. To cancel object selection, re-select it.



If you need to add all faces of a surface or a body to the current surface set, select the body or surface in the Design Tree.



It is impossible to construct a coordinate plane intersection curve.

The ghost of the intersection curve will display on the screen on selection of intersection objects.



An intersection curve may consist of one or several continuous segments - contours. Select the contours required for construction (see Section 2.5.4.15.1 on p. 414).

To change the intersection objects click the **Select Other** button on the Advanced Control panel and re-select the objects.

The controls contained on the **Properties** tab of the Properties bar enable changing the name and the color of the intersection curve, and set the parameters of its display (display parameters are detailed in Section 2.5.4.1 on p. 361).



To finish the construction of the intersection curve, click the **Create Object** button in the Special Control Panel.



The intersection curve will appear in the Model window and the intersection icon will appear in the Design Tree.

2.5.4.24. Contour



You can create a contour using already existing objects – 3D curves, edges or sketch lines.

To create a contour, call the **Contour** command of the **3D Lines** toolbar. The command is accessible if there is at least one curve, surface, body or sketch in the document.

Select contour type using the **Type** group of switches of the Properties bar or context menu commands.

- ▼ **Automatic Type Detection,**
- ▼ **Arbitrary Contour,**
- ▼ **Contour on edge,**
- ▼ **Sketch Contour.**



The activation of the switch **Automatic Type Detection** means that contour type will be automatically determined by the system after selecting the curves.

Specify objects which comprise the contour of selected type: curves, sketch lines, edges of bodies and surfaces, other previously created contours.



When specifying objects, conditions of continuity of the contour to be created and absence of self-intersections should be met.

For detailed description of contour types and object selection, refer to Section 2.5.4.24.1 on p. 438.



Objects can be selected before calling the command.

The ghost contour will be displayed. Start vertex of the contour is marked with an arrow. Reselecting of the object cancels its selection together with all other objects towards the end vertex of the contour.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.

The names of selected objects are displayed in Table **List of Objects** on the Property Bar.

You can create fillets with permanent or variable radius in the contour's angles (see Section 2.5.4.24.2 on p. 439). You can also create a contour by the curve's equidistant (see Section 2.5.4.24.3 on p. 440).



To change objects used for contour construction, click the **Reselect** button on the Special Control Panel and re-select objects.

Controls on the **Properties** tab of the Property bar make it possible to adjust the contour's name, color and display parameters. For detailed information on display parameters of curves, see Section 2.5.4.1 on p. 361.



The **Starting/Ending vertex** group of switches makes it possible to change the starting vertex of the contour.

The **Hide source curves** option controls visibility of source objects of the contour after its creation. If it is enabled, curves by which the contour is built will be hidden automatically after the contour's creation.



To finish construction, click the **Create Object** button on the Special Control Panel.

Created contour will appear in the model window, and its icon will be displayed in the Design Tree.



Note that the contour is associated with objects which had been used for its construction, therefore, the contour form changes when forms of these objects change.

2.5.4.24.1. Selecting contour type

You can set following contour types using the **Type** group of switches of the Properties bar or context menu commands.



Arbitrary Contour— a contour composed of 3D curves, edges, sketch lines, other contours, separate segments of the multi-segment curves.

To create an arbitrary contour, select the curves that shall be included in it. For object selection refer to Section 2.1.4.1 on p. 130.



Contour on edge— a contour consisting of one face edges.

To create a contour on face, select its edge that shall be included in contour. One of the faces, to which the edge belongs, will be highlighted. To select an adjacent face instead of this one, click the adjacent face by the mouse.

Specify all necessary settings for the edge's contour of the selected face.



Sketch contour— a contour consisting of one sketch lines.

Specify the sketch lines in the Model window or the sketch in the Design Tree. If the sketch is selected in a Tree, the contour will comprise all its lines; therefore, they should be connected one by one without self-intersections.



Automatic de of type is the default method of contour creation which does not require special selection of type. When the **Type Auto Determination** switch is active, you can select any curves. After the contour creation, its type is determined by the system automatically. To verify this, enter the editing mode of the contour created with the **Type Auto Determination** switch enabled. Enabled switch indicates the contour type.

While selecting, you can switch between types. Depending on which objects are specified for contour creation, the **Sketch Contour**, **Contour on Face** switch or both of them may be unavailable.



After *sketch contour* or *contour on face* creating, you can add objects that do not belong to this sketch or face. To do this, click the **Arbitrary Contour** switch and select objects.

Contour type is important while executing operations. For example, during extrusion operation, selection of a contour type affects the result of the operation (see Section 2.3.2.3.1 on p. 154).

2.5.4.24.2. Creation of a contour with fillets



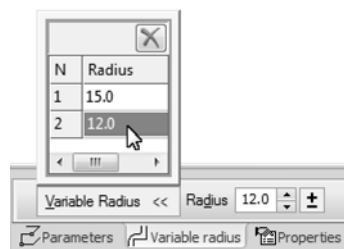
In order to create fillets in a contour's corners, enable the **Passing with fillet** switch in the **Angle passing** group of the **Parameters** on the Properties bar. Set the fillet's *permanent* radius value in the **Radius** field.

Fillets with the same radius will be displayed in the contour's ghost in the model window. In the first fillet, there will be a handle for controlling the value of the radius. When moving the point with the mouse, all fillets in the ghost will be changing.

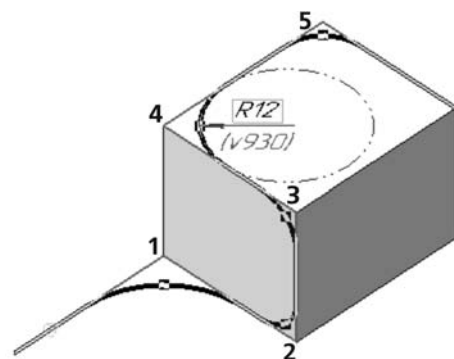
In order to set a *variable* radius, i.e. specific radius value in each vertex of the contour, go to the **Variable radius** tab with the **Passing with fillet** switch enabled.

The tab contains the respective bar with the table of parameters: numbers of points and values of radii in them (Fig. 2.5.62, a). If the points are not specified, the table will be empty.

The contour's ghost in the model window displays handles of all radii, numbered from the contour's beginning (Fig. 2.5.62, b).



a)



b)

Fig. 2.5.62. Example of creating a contour with the variable radius of fillets: a) specifying the value of the radius in the table; b) displaying fillets in the contour's ghost

Use the cursor to specify the points the radius of which should be changed. The selected points are displayed in the **Variable radius** table.

Specify the fillets' radii values in these points using any method – by moving handles using the mouse or by setting values in the table.



In order to change the radius value in the table, click or double-click the required cell. Upon a single click, you can enter values by typing from the keyboard, upon a double click, you will be able to enter values by typing from keyboard or selecting by the stepper button.



Fillets are displayed in the contour's ghost in the model window.

You can delete a point from the table using the **Delete** button located above the list. Deletion of a point from the table means that a fillet with the constant radius will be built in it.

2.5.4.24.3. Creation of a contour by the curve's equidistant

You can create a contour by the equidistant of the specified curve when using the **Contour** command.



In order to set parameters of the equidistant, click **Curve's equidistant** button on the Special Control panel.

The system will start creation of the equidistant curve which is similar to the process enabled by the command **Curve's equidistant** (see Section 2.5.4.17 on p. 417). Select the equidistant creation method, then specify the required parameters (enter the values or specify the objects). All changes are displayed in the curve ghost in the model window.



Curves can be specified before starting the process of the equidistant construction.



Upon completion of the equidistant parameterization click the **Create Object** button on the Special Control Panel.

The system will return to execution of the **Contour** command.

The name of the equidistant curve will be displayed in the **List of objects** table in the Property bar as an object for creation of the contour.



You can create angle passing with fillets both in the equidistant and in the contour. Creation of fillets in contours is more convenient, since arbitrary radii values can be set (see Section 2.5.4.24.2 on p. 439).

One contour by equidistant can be created by calling the **Contour** command once. In order to add other curves to the contour, call the **Contour** command again and specify the objects.



Note that the equidistant constructed during creation of the contour will not be displayed in the Design Tree as a standalone object. It will be saved inside the *Contour* object for which it had been created.

In order to change parameters of the equidistant as part of the contour, enter the contour edit mode and click the **Curve's equidistant** button on the Special Control panel. The system will

start the process of editing the equidistant which is the same as the process of its construction. Change parameters of the current method of equidistant creation or choose another method and re-create the curve. Once the editing is finished, click the **Create Object** button to exit the equidistant editing process.

2.6. Surfaces

2.6.1. Creation of surfaces

The following surfaces can be created in the KOMPAS-3D model file:

- ▼ extrusion surface,
- ▼ revolution surface,
- ▼ lofted surface,
- ▼ Sweep;
- ▼ Patch;
- ▼ Ruled,
- ▼ Surface by curve grid;
- ▼ Surface along a point grid,
- ▼ Surface along point formation,
- ▼ Equidistant,
- ▼ Blend surface.

The user may import the surfaces saved into a SAT, IGES or STEP AP203 file into the surface model.

The following surface operations are available:

- ▼ Trimming;
- ▼ Splitting,
- ▼ Extension;
- ▼ Knitting.

Besides that, you can modify or remove individual faces of surfaces and bodies.

Commands to work with surfaces are located in menu **Operations — Surface**. The respective command buttons are located on the **Surface** toolbar (Fig.2.6.1).



Fig. 2.6.1. **Surfaces** toolbar

All of the above-mentioned options for creating surfaces and working with them are described in this section. Besides that, the following operations with surfaces are possible:

- ▼ Scaling (see Section 2.3.8.7 on p. 204),
- ▼ Position changing (see Section 2.3.8.8 on p. 205),
- ▼ saving a new part in file, see Section 2.11.2.3 on p. 689.

Surfaces can be used, for example, to cut a model's part or as an object, which is the limit for extrusion.



During creation, the surface should be integer, i.e. it should not comprise parts. Otherwise, the execution of operation is impossible.

2.6.1.1. Imported Surface

To import surface or surfaces to a KOMPAS-3D model file the following files may be used.

- ▼ File with *sat*, *igs*, *stp* or *step* extension — free surfaces and surfaces of the bodies contained in this file will be inserted into the current model.
- ▼ The coordinate file with *txt*, *xls*, or *ods* extension (see Chapter VI) – the polyface surface will be built in the current model using the coordinates from the file. Every face of surface has triangle shape.



To perform the import, use the **Imported Surface** command.

After calling the command, a standard Select File dialog appears, where you need to select the desired file.



When reading in the **.igs* file containing no topology, a prompt will be displayed for knitting surfaces. An affirmative answer implies that the imported surfaces will be knitted wherever possible.



Once the import has been finished, icon of the imported surface or several icons of imported surfaces will appear in the Design Tree.

The coordinate system of the imported surface or surfaces will be the current coordinate system – absolute or local (see Section 2.8.3.3 on p. 597 for more details about selection of the current coordinate system).



If the coordinate system of an imported surface is not absolute but local, then you can change the position of the imported surface in the model by changing the position of the local coordinate system relative to the absolute coordinate system.

For detailed description of coordinate systems of objects, refer to Section 2.8.3.4 on p. 597.

If an assembly model is stored in the selected *.sat*, **.igs*, **.stp* or **.step* file, the reading will result in creation of parts with imported surfaces in the current model. At saving of the model the files of these parts shall be saved to disk. Those are placed in the same folder as the model.

2.6.1.2. Extrusion Surface

Extrusion surface is created by sweeping section along the straight-line guide in either or both sides at the specified distance.

A face, sketch, edge or 3D curve (including contour constructed by sketch lines, by face edges or arbitrary¹) may be used as a section of extrusion surface.

1. It means that has **Contour on Face**, **Sketch Contour** and **Arbitrary Contour** type, see Section 2.5.4.24.1 on p. 438.

During extrusion of the face, closed sketch, closed **Sketch Contour** or **Face Contour**, the construction of a closed surface is possible (Fig.2.6.2). If the section is open, then surface closing is impossible.

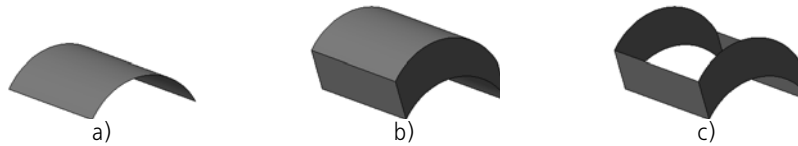


Fig. 2.6.2. Face extrusion: a) section (cylindrical face), b) closed surface, c) open surface

If section is a flat face, sketch, contour constructed by sketch lines or flat face, and it is extruded at the direction perpendicular to it, sloping to side faces of feature is possible (Fig.2.6.3).



Fig. 2.6.3. Slope of side faces of extrusion surface: a) closed, b) open



A sketch used as the section of extrusion surface should satisfy the requirements listed in Section 2.6.1.2.2.

During edge or 3D curve extrusion (including contour of **Arbitrary** type), close and slope of the surface's faces are impossible. Example of 3D curve extrusion is shown in the Figure2.6.4.



Fig. 2.6.4. Extrusion element with a spiral section

2.6.1.2.1. Construction of Extrusion Surface



1. Call the **Extrusion Surface** command.

The surface creation operation starts. The operation parameters management interface items will appear on the Property bar.

2. Specify the operation parameters:

- ▼ **section** — in the same way as in the extrusion operation, see Section 2.3.2.3.2;
- ▼ **extrusion direction** — in the same way as in the extrusion operation, see Section 2.3.2.4;
- ▼ **extrusion depth** — in the same way as in the extrusion operation, see Section 2.3.2.5;

- ▼ **slope angle** — in the same way as in the extrusion operation, see Section 2.3.2.6.



To set up such parameters as slope angle, extrusion direction and depth, you can use handles (handles are detailed in Section 2.1.4.2.2 on p. 138).

All parameter values are displayed on the screen as a surface ghost.

3. If a face, closed sketch, closed **Contour on face** or **Sketch contour** are used as section, the **Closed surface** option will become available. Enable it, if you need to add faces to a face surface, that correspond to the initial and the final position of the section (see Fig.2.6.2, b).



4. click the **Create button** to fix surface.

Surface shall appear in the model window and the corresponding icon — in the Design Tree.

2.6.1.2.2. Requirements to the Sketch of Extrusion Surface

- ▼ Sketch objects can be a part of one or several contours.
- ▼ If there is only one contour, it can be either open or closed.
- ▼ If there are several contours, then:
 - ▼ all of them must be closed.
 - ▼ one of them must be the outer one and the rest must be nested in it; allowed level of nesting — 1.



If a sketch contains nested contours, the correct construction of a surface is possible only if it is closed, i.e. the **Closed Surface** option is on.

2.6.1.3. Revolution Surface

Revolution feature is created by rotation of section about the axis in either one or both sides at the specified angle.

A face, sketch, edge or 3D curve (including contour constructed by sketch lines, by face edges or arbitrary¹) may be used as a section of revolution surface

During rotation of the face, closed sketch, closed **Sketch Contour** or **Face Contour**, the construction of a closed surface is possible (Fig.2.6.5).

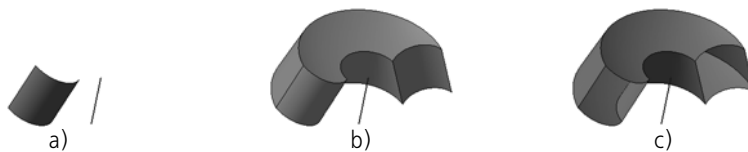


Fig. 2.6.5. Face Rotation: a) section (cylindrical face) and axis, b) closed surface, c) open surface

If the section is open, then surface closing is impossible.

1. It means that has **Contour on Face**, **Sketch Contour** and **Arbitrary Contour** type, see Section 2.5.4.24.1 on p. 438.

During edge or 3D curve rotation (including contour of **Arbitrary** type), closing a surface is also impossible. Example of 3D curve revolution is shown in the Figure 2.6.6.



Fig. 2.6.6. Rotation element with a spiral section

If the section is an open sketch or an open contour built by the sketch or flat face's edges and rotates around the axis within the section's plane, the construction type selection is available—toroid or spheroid (Fig.2.6.7).



Fig. 2.6.7. Spheroid and toroid construction (section—sketch—marked with a thickened line): a) spheroid, b) toroid



A sketch used as the section of revolution surface should satisfy the requirements listed in Section 2.6.1.3.2.

2.6.1.3.1. Construction of Revolution Surface



1. Call the **Revolution Surface** command.
The surface creation operation starts. The operation parameters management interface items will appear on the Property bar.
2. Specify the operation parameters (in the same way as in the revolution operation):
 - ▼ **Section** (see Section 2.3.3.3 on p. 168).
 - ▼ **revolution axis** (see Section 2.3.3.4 on p. 169);
 - ▼ **creation type** (see Section 2.3.3.5 on p. 170);
 - ▼ **revolution direction** (see Section 2.3.3.6 on p. 171);
 - ▼ **revolution angle** (see Section 2.3.3.7 on p. 171).



To set up such parameter as angle of revolution, you can use handles (handles are detailed in Section 2.1.4.2.2 on p. 138).

All parameter values are displayed on the screen as a surface ghost.



3. When using faces, closed sketch, closed **Sketch contour** or **Face contour**, the **Closed surface** option is available. Enable it, if you need to add faces to a face surface, that correspond to the initial and the final position of the section (see Fig.2.6.5, b).
4. Click the **Create button** to fix surface.

Surface shall appear in the model window and the corresponding icon — in the Design Tree.

2.6.1.3.2. Requirements to Sketch of Revolution Surface

- ▼ A sketch may contain one or several contours.
- ▼ If there is only one contour, it can be either open or closed.
- ▼ If there are several contours, all of them must be closed.
- ▼ If there are several contours, one of them must be the outer one and the rest must be nested in it.
- ▼ Only one level of nesting is permitted for contours.



If a sketch contains nested contours, the correct construction of a surface is possible only if it is closed, i.e. the **Closed Surface** option is on.

Besides the section, sketch may also contain the rotation axis. In this case, additional requirements shall be met:

- ▼ the axis should be drawn in a sketch as a line segment with *Centerline* line style or as an *Centerline* object type,
- ▼ all contours must lie on the same side of the revolution axis.
- ▼ no contour should intersect the axis (or its extension),
- ▼ some contour points may be on the axis (or its extension),
 - ▼ for closed contours – any of their points and/or segments,
 - ▼ for open contours – only their outermost points.

2.6.1.4. Swept Surface

Swept surface is created by sweeping section along the path.

If section is closed, the construction of a closed surface is possible (Fig.2.6.8). If the section is open, then surface closing is impossible.

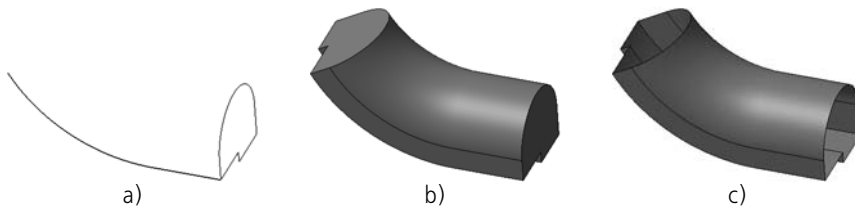


Fig. 2.6.8. Swept surface: a) section and path, b) closed surface, c) open surface

Surface section must be displayed in the sketch; any curve or chain of curves can be used as a path. The requirements to the section sketch and surface path are the same, as for the sketch and the path of the swept feature (see Sections 2.3.4.3.1 on p. 174 and 2.3.4.4.1 on p. 175).

Do the following to create a swept surface in the model.



1. Call the **Swept Surface** command. The command is available if there is at least one sketch in the model.

The surface creation operation starts. The operation parameters management interface items will appear on the Property bar.

2. Specify the operation parameters (in the same way as in the sweep operation):

- ▼ **Section** (see Section 2.3.4.3 on p. 174).
- ▼ **path** (see Section 2.3.4.4 on p. 175);
- ▼ **type of section movement** (see Section 2.3.4.5 on p. 175);

All parameter values are displayed on the screen as a surface ghost.

3. When using closed section, the **Closed Surface** option is available. Enable it, if you need to add flat faces to a face surface, that correspond to the initial and the final position of the section (see Fig.2.6.8, b).



4. Click the **Create button** to fix surface.

Surface shall appear in the model window and the corresponding icon — in the Design Tree.

2.6.1.5. Lofted Surface

Surface by sections is formed by joining several sections of arbitrary form and position. Extreme section can be a point.

If sections are closed, the construction of a closed surface is possible (Fig.2.6.9). If the sections are open, then surface closing is impossible.

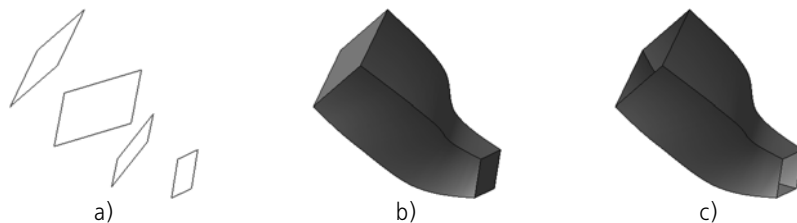


Fig. 2.6.9. Surface by Sections: a) sections, b) closed surface, c) open surface

If necessary, you can use a centerline during element creation by sections (Fig.2.6.10).

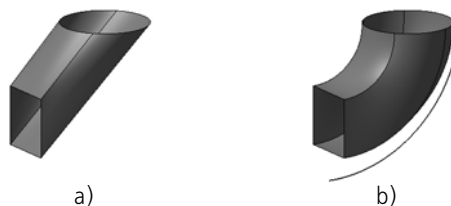


Fig. 2.6.10. Surface by Sections: a) without centerline, b) with centerline

Surface sections must be displayed in the sketch; any curve or chain of curves can be used as a centerline. The requirements to the section sketches and centerline are the same, as for the sketches and the centerline of the lofted feature (see Sections 2.3.5.3.1 on p. 179 and 2.3.5.4.1 on p. 180).

Do the following to create a surface by sections in the model.



1. Call the **Surface by Sections** command. The command is available if there are at least two sketches in the part.

The surface creation operation starts. The operation parameters management interface items will appear on the Property bar.

2. Specify the operation parameters (in the same way as in the operation by sections):

- ▼ **Section** (see Section 2.3.5.3 on p. 178).
- ▼ **Centerline** (see Section 2.3.5.4 on p. 179);
- ▼ **Creation method** (see Section 2.3.5.5 on p. 180).

All parameter values are displayed on the screen as a surface ghost.

3. When using closed sections, the **Closed Surface** option is available. Enable it, if you need to add flat faces that of the same shape as the initial and final sketched sections to a surface (see Fig.2.6.9, b).



4. Click the **Create button** to fix surface.

Surface shall appear in the model window and the corresponding icon — in the Design Tree.

2.6.1.6. Surface Along a Point Grid

Surface created by the point grid — is a spline surface constructed by points that are conventionally grouped in a grid, i.e. the points are arranged in several rows with the same number of points in each row. Merge points are not allowed.

Direction U is the direction of adding points to create a surface. Direction V — is the direction to add rows. Thus, the points of the grid constitute rows in directions U and V.

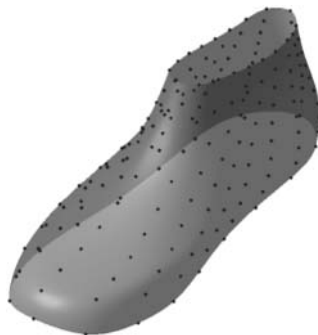


Fig. 2.6.11. Surface Along Point Grid

2.6.1.6.1. Creating Surface Along Point Grid



To draw a surface with specified end points, click **Surface by Grid of Points**. Control elements for joining curves will appear on the Properties Bar (Fig.2.6.12).



Fig. 2.6.12. Property toolbar if LCS is created by the basic method



Select the surface type using the **Type** group of switches: **By Points** or **By Poles**. A surface by points passes directly through the grid points. A surface by poles passes at a distance from the grid points as defined by their weight. By default, weight of the points is 1; if required, the weight can be changed through the surface edit mode (see Section 2.6.1.6.3 on p. 452).

Specify the surface points (see Section 2.6.1.6.2),

You can define the thread name and color on the **Properties** tab of the Property bar. The segment is constructed parallel to the selected axis of the polyline's coordinate system (for object's coordinate system, refer to Section 2.8.3.4 on p. 597).

If you want to check surface for self-crossing, enable the **Check self-crossing** option.



When you have specified points and surface parameters, click the **Create Object** button on the Special Control panel.



The **Create object** button is accessible if the even number of point rows is input, starting from two.



The created surface will appear in the model window, and the surface icon — in the Design Tree.

If the **Self-Intersection Check** option was enabled and returned a positive result the surface will be marked in the Design Tree with an exclamation mark in a red circle which indicates errors. For more information about self-intersection of surface refer to Section 2.6.1.6.5.

2.6.1.6.2. Specifying Points for Surface Along Point Grid

To specify surface points, select individual point objects in the Design Tree or model window (the list of point objects can be found in Table 2.1.7 on p. 131), group of points (see Section 2.5.3 on p. 348 for more information about groups of points) or array of points (see Section 2.7 on p. 509 for more information about arrays) in the Model Tree or model window.

Selecting objects with the **Associate** option enabled will build an associative relation between the surface points and the objects.



You can create a point by clicking the **Point Drawing** button on the Special Control Panel (for details see Section 2.5.2.11.1 on p. 347).

Specify and create points in the order of their arrangement in a row.

Number of the next point and total number of grid points are displayed in the corresponding fields of the Property Bar.



Having preset points of the first row, click the **Finish the Row** button. This done, the button disappears from the Property Bar. Number of points in the second and following rows is accepted equal to the number of points in the first row.



If you have specified a group of surface points or an array of parallelogram grid points with more than 1 instance on both axes, the number of rows and points will be determined automatically.

Select or create points for other rows.

The screen will show the ghost of the surface being created, the grid ghost and the arrows of directions U and V.



To cancel selection of one or several final points click the **Cancel Selection button** on the Special Control Panel. To specify another reference point, click the **Select Other** button.



If a different number of rows is set (beginning with two) the **Edit** button will be available on the Advanced Control panel. Enables surface edit mode (see Section 2.6.1.6.3 on p. 452).



If none of the grid points is specified (i.e. immediately after running the command or clicking the **Re-select** button) the following buttons are available on the Advanced Control panel:



- ▼ **Read from File** which enables acquiring point coordinates saved into a file (more information about the data file can be found in Appendix VI); the systems interprets the numbers saved in the file as point coordinates in the Cartesian coordinate system which coincides with the surface coordinate system (for more information about the object coordinate system see Section 2.8.3.4 on p. 597),



- ▼ **Surface Grid** which allows to construct a surface using the grid which approximates any of the existing ones to the surface models (see Section 2.6.1.6.4 on p. 453).

2.6.1.6.3. Surface Editing Mode Along Point Grid



To switch to the editing mode click the **Edit** button on the Advanced Control panel. Property toolbar contains controls shown in Figure 2.6.13.

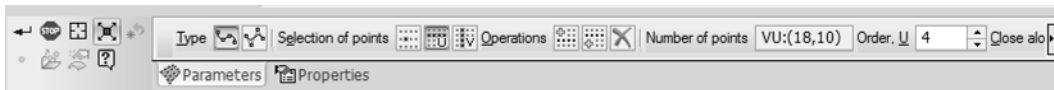


Fig. 2.6.13. Property bar in Edit Surface by Point Grid mode



The system automatically switches to the surface editing mode on running the edit command for an earlier created surface.

In this dialog you can

- ▼ change position and weight of points, delete and add rows,
- ▼ change surface order,
- ▼ enable and disable surface closing.

These methods are detailed below.

Moreover, the **Type** group of switches which controls the surface type and the **Self-Intersection Check** option which verifies the surface for self-intersections are available in the surface editing mode.

All updates shall be displayed on the LCS ghost in the model window.

To continue designation of the grid points after editing the surface release the **Edit** button. To stop surface creation, click the **Create Object** button.



Editing Points and Rows

Select the edit method using switches of the **Point Selection** group:



- ▼ Enable switch **By one point** to select points one by one. Select a point to edit, click on it in the model window.

- ▼ To set a new point position, specify a new point object or drag the or select other point construction method (for more details refer to paragraph 3, Section 2.5.2.11.2 on p. 347).

- ▼ To change a point weight (for the **By pole** surfaces), set the required value in the **Weight** field.



- ▼ To select the required row of points, enable either **U direction row** or **V direction row** switch. To specify a row to be edited, click any point of the same row.



- ▼ To add a new row in the same direction, enable either **Insert before the selected row** or **Insert after the selected row** in the **Operations** group on the Property bar. A new row will appear in the point grid. Points in this row are created **By Coordinates (XYZ)**.



- ▼ To delete a row, enable switch **Delete the Selected Row** in the **Operation** group.



- ▼ To change a point weight (for the **By pole** surfaces), set the required value in the **Weight** field. Each point in the row will acquire the specified weight.

Surface order in the U and V direction

The surface order is one of the surface parameters which define its shape. For details about the order of the spline curves and surfaces, see Section 2.5.1.4 on p. 329.

Surface order parameters are set in fields **Order, U** and **Order, V**.

Surface Closing



You may close surface either in U or V direction, or in both directions. To do this use switches **Closed in U** and **Closed in V** are provided.



Closing in either direction is possible if there are three or more points in the respective direction.

2.6.1.6.4. Constructing Point Grid Along Existing Surface



A point grid can automatically be created based on the existing surface model. The following can be used as a surface:

- ▼ solid or surface face,
- ▼ auxiliary or coordinate plane;

To do this, click the **Edit Points** button on the Special Control panel, and then “drag” polyline vertices with the mouse.

The selected surface will be approximated by the spline surface on the grid. The surface' ghost and point will appear in the model window.

- ▼ If the selected surface is an auxiliary or coordinate plane, the approximating surface coincides with the square which displays this plane in the model space.
- ▼ If a face has been specified as a surface, the approximating surface will coincide with the face' theoretical surface (see Section 2.5.1.3 on p. 328). Thus, holes of the selected face will be ignored; in some cases the approximating surface borders do not coincide with contours of the selected face: the surface overlaps the edge. For example, if the selected face is a circle, then the approximating surface will be a square positioned next to the circle.



The approximating surface is always the spline surface, and its borders are spline curves even if a plane or flat face with straight edges is selected for approximation.

2.6.1.6.5. Self-Intersecting Surface

Self-intersection of a surface as a mathematical object is not an error. At the same time self-intersection of a part face is a modeling error since such part cannot be manufactured.

Due to the above it is impossible, for example, to define width using the command of the same name (see Section 2.3.6 on p. 184) since this operation will invariably create a body in error. However, a surface segment devoid of self-intersections can be widened by separation from the rest of the surface by means of trimming (see Section 2.6.2.1 on p. 469).

One more example: extrusion to a surface (see Section 2.3.2.5.4 on p. 159) is impossible if the element section is projected onto the surface at the part where it self-intersects. If the section projects onto the surface part devoid of self-intersection the program will proceed with extrusion.

Since in most cases surfaces are built for further use as body faces such commands as **Surface Along Point Grid** and some others contain the option of checking for self-intersection. If checking is enabled, any surface containing self-intersections will be marked in the Design Tree as surface in error. The need to correct such an error, as illustrated above, depends on which part of the surface is intended for further work.

2.6.1.7. Surface Along a Point Formation

Surface constructed with the aid of the point layer is a spline surface or a polyface surface created by points which form a space layer, i.e. they are arranged in such a manner that the height of their circumscribing rectangle is lower with respect to their length and width.

During execution of the **Surface Along Point Formation** the system interprets the formation as a point grid which is used for surface construction. Therefore, when creating a spline surface this command can be used as an equivalent of the **Surface Along Point Grid** (see Section 2.6.1.6 on p. 450) which does not require manual specification of points. If the automatic point grid recognition result is unsatisfactory, use the **Surface Along Point Grid** command.

The **Surface Along Formation** command yields better results if the formation points form a regular grid with rectangular spaces.



Fig. 2.6.14. Point formation and the surface built along it

2.6.1.7.1. Surface Construction Along Point Formation



To construct a surface along a point formation run the **Surface Along Point Formation** command. Control elements for joining curves will appear on the Properties Bar (Fig.2.6.15).

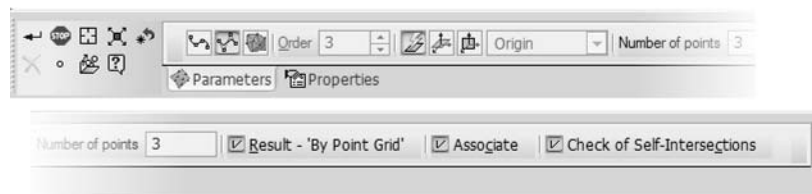


Fig. 2.6.15. The Properties bar during creating a surface along a point formation

Select the surface type using the **Type** group of switches:



▼ **By points.** A spline surface that passes through the grid points.



▼ **By poles.** A spline surface that passes at a certain distance from the points. The distance is defined by the system and cannot be modified.



▼ **Polyface.** A surface that passes through the grid points. Every face of surface has triangle shape.

Specify the surface points (see Section 2.6.1.7.2),

The **Order** field enables setting the surface order. For details about the order of the spline curves and surfaces, see Section 2.5.1.4 on p. 329.

The switches of the **Method** group are used to select the method of grid points recognition. The method of grid recognition determines the surface shape (see Section 2.6.1.7.3).

To force the command to create a surface along a point grid (not a surface along a point formation), enable the **Result - 'Along Point Grid'** option.

If you want to check surface for self-crossing, enable the **Check self-crossing** option.



If the surface is of a polyface type, the **Order** field, options **Result-"By point grid"** and **Check for self-intersections** will not be available.

You can define the thread name and color on the **Properties** tab of the Property bar. the segment is constructed parallel to the selected axis of the polyline's coordinate system (for object's coordinate system, refer to Section 2.8.3.4 on p. 597).



When you have specified points and surface parameters, click the **Create Object** button on the Special Control panel.



The **Create Object** button becomes available if the user specifies three or more points.



▼ Surface Along Point Formation icon;



▼ Surface Along Point Grid icon (if the **Result - 'Along Point Grid'** option was enabled).

If the **Self-Intersection Check** option was enabled and returned a positive result the surface will be marked in the Design Tree with an exclamation mark in a red circle which indicates errors. For more information about self-intersection of surface refer to Section 2.6.1.6.5.

2.6.1.7.2. Specifying points for a surface along a point formation

To specify surface points, select individual point objects in the Design Tree or model window (the list of point objects can be found in Table 2.1.7 on p. 131), groups of points (see Section 2.5.3 on p. 348 for more information about groups of points) or arrays of points (see Section 2.7 on p. 509 for more information about arrays) in the Model Tree or model window. Merge points are not allowed.

Selecting a point object or a group of points with the **Associate** option enabled will result in creation of an associative relation of the point with these objects.



You can create a point by clicking the **Point Drawing** button on the Special Control Panel (for details see Section 2.5.2.11.1 on p. 347).

The surface ghost will appear after specifying the third point. The points are numbered in the order they are preset.



The **Read from File** button enables acquiring the coordinates of points saved in a file (data file is detailed in Appendix VI). The system interprets the numbers saved in the file as point coordinates in the Cartesian coordinate system which coincides with the surface coordinate system (for more information about the object coordinate system see Section 2.8.3.4 on p. 597). The points imported from a file are added to the already specified points with the ghost reconstruction.



To cancel selection of one or several final points click the **Cancel Selection** button on the Special Control Panel. To specify another reference point, click the **Select Other** button.



If at least one point of a surface is specified, the **Edit** button becomes available on the Advanced Control panel. Enables surface edit mode (see Section 2.6.1.7.4 on p. 457).



2.6.1.7.3. Recognition of Point Grid

There are three methods of point recognition available. All the methods imply searching for grid cells — directly in space or in the plane. After all the points are projected onto a plane the system will search for cells formed by the point projections.

Cells shape depends on the type of the surface: Cells of surface **By Points** or **By Poles** are quadrangular, cells of **Polyface Surface** — triangular.

If the search returns any cells on the surface they are used for allocating cells in space.

Triangular cells, unlike quadrangular, can always be found.

If the search fails to find any rectangular cells in space or on the surface the grid is built in such a way that each row of one direction passes through a formation point whereas in the second direction the grid has two rows.

The **Method** group of switches is used to select the method of recognizing the grid formed by the formation points.



- ▼ If the **Automatic** recognition method is selected the system first searches for grid cells in the space. If the search returns no results, the system continues to search for cells on the surface the position of which is determined automatically based on the relationship of the formation points.



- ▼ If the **In Coordinate System Plane** option is enabled, the system will search for cells in XY plane of the coordinate system specified by the user. If this switch is enabled the list of coordinate systems becomes accessible on the Properties bar.



- ▼ If the **In Screen Plane** method is selected, the system will search for cells in the plane parallel to the screen - i.e. the surface shape will depend on the model position relative to the observer. Enabling this switch reveals the **Fix Position** option on the Properties bar. Enabling this option fixes the surface in the form corresponding to the current position of the model. With the option disabled the surface will be reconstructed each time after changing the model orientation. Note that the surface form (especially, polyface) can change sharply in case of a relatively small change of the model position.

The following order of point grid detection is recommended.

1. Use **Automatic** determination method
2. If the automatic recognition result is unsatisfactory, use the **In Coordinate System Plane** and **In Screen Plane** to pick up suitable plane for points projecting.
3. If you do not get the required form of the surface using these methods, edit the point layer: create additional points (see Section 2.6.1.7.2), exclude or move points (see Section 2.6.1.7.4).

2.6.1.7.4. Surface Editing Mode Along Point Formation



To switch to the editing mode click the **Edit** button on the Advanced Control panel.

Property toolbar contains controls shown in Figure 2.6.13.

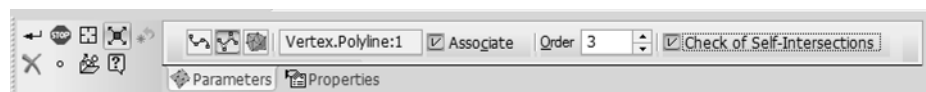


Fig. 2.6.16. The Properties bar while editing a surface along a point grid



The system automatically switches to the surface editing mode on running the edit command for an earlier created surface. In this case the Properties bar differs from the one shown on Fig. 2.6.13 by the lack of the **Result - Along Point Grid** option, i.e. transforming an already existing surface created along a point formation into a surface along a point grid is impossible.

The editing mode enables changing the position of points as well as deleting them. To specify the point to be edited click it with the mouse in the model window.

- ▼ To set a new point position, specify a new point object or drag the or select other point construction method (for more details refer to paragraph 3, Section 2.5.2.11.2 on p. 347).



- ▼ To delete a point click the **Delete Point** on the Advanced Control panel.

Other controls accessible in the surface editing mode function identically to the surface creation mode (see Section 2.6.1.7.1).

All updates shall be displayed on the LCS ghost in the model window.



To continue designation of the grid points after editing the surface release the **Edit** button. To stop surface creation, click the **Create Object** button.

2.6.1.8. Surface Along a Curve Grid

The surface along a curve grid is a spline surface built along two intersecting families of curves. The curve intersection points are called **grid nodes**.

All curves of the first family are considered U direction curves, while all curves of the second family are considered V direction curves. Points can be specified for one direction as one or two end curves.

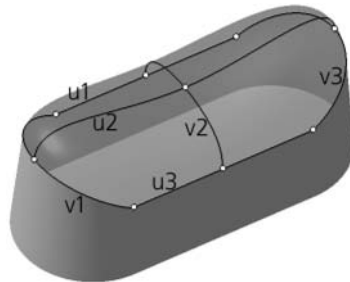


Fig. 2.6.17. Surface Along Curve Grid.
u1, u2, u3 are the curves of direction U whereas v1, v2, v3 are the curves of direction V.
The grid nodes are marked with circles

2.6.1.8.1. Curves and grid points

Curves and grid points collocation requirements:

- ▼ Each curve of one direction shall have one common point with each curve of other direction.
- ▼ Collocation of curves having different directions is inadmissible.
- ▼ Curves of the same direction shall be either closed or opened.
- ▼ The points can be specified instead one or both end curves in any direction. In this case, all curves of other direction shall pass through the point(s) specified for this direction.

The curves forming the grid may be represented by any single-segment spatial curves, edges and sketch lines, whereas points may be represented by any point objects.

2.6.1.8.2. Construction of Surface Along Curve Grid



To create a surface by the curve grid, call the **Surface by Curve Grid** command. Control elements for joining curves will appear on the Properties Bar (Fig.2.6.18).

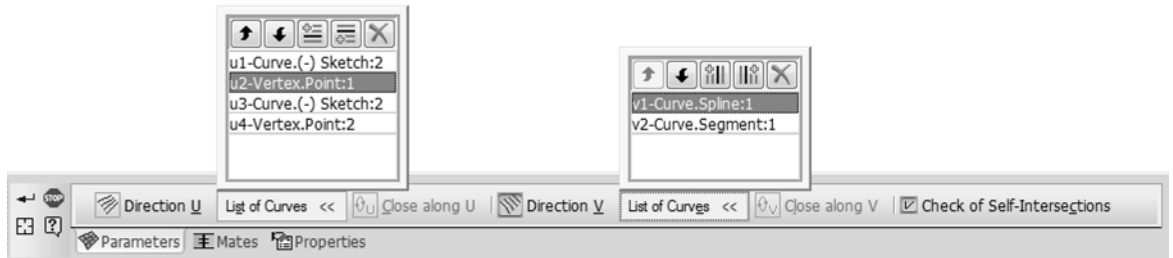


Fig. 2.6.18. Property Bar for constructing surface by curve grid



Switch **Direction U** is enabled automatically. Select objects for U direction— curves and curves and points with reference to their sequence order on the grid. Minimum two curves, or one curve and a point, or one curve and two points shall be specified for this direction. The corresponding direction of creation can be selected from the **Direction** list in the Property bar. The model window displays the surface ghost.

For V direction you may select no curve, or select one or several curves. If no points are selected for U direction, it is possible to specify points for V direction.



If one curve is preset for V direction, such curve shall pass through end (start of final) points of all U curves.



To select the V direction objects, enable switch **V direction** on the Property bar and select the required objects with reference to their sequence order on the grid. They will be added to the **V direction curves** list.



The control buttons of the list elements are located above the curve lists.



- ▼ To change position of a curve in the list select it and click button **Move up** or **Move down**.



- ▼ To add a curve before or after the specific curve in the list, select the curve in the list, click the **Insert button before the curve** or **Insert after the curve**, then specify the curve to be added.



- ▼ To remove any curve from the list, select it and click **Delete**.



If you need to clear both curve lists, click the **Reselect** button on the Special Control panel.



Switches **Close by U** and **Close by V** allow to close the surface. The surface becomes closed in either direction if all curves of the same direction are closed.



If the surface can be closed in any direction, the corresponding switch is accessible.



If you want to check surface for self-crossing, enable the **Check self-crossing** option.

If the created surface links up with an existing one, you may preset mating conditions for the created surface. The mating conditions can be preset in the **Mating** tab of the Property bar (see Section 2.6.1.8.3).

All surface modifications are displayed on its ghost in the model window.

You can define the surface name and color on the **Properties** tab of the Property bar.



To finish construction of curves and save the preset surface properties, click the **Create Object** button on the Special Control Panel.



The created surface will appear in the model window, and the surface icon — in the Design Tree.

End curves or points are boarders for the created surface. If no (or one) curve is selected for V direction, the surface boarders (of one of the boarders) in this direction will be determined in automated mode.

If the **Self-Intersection Check** option was enabled and returned a positive result the surface will be marked in the Design Tree with an exclamation mark in a red circle which indicates errors. For more information about self-intersection of surface refer to Section 2.6.1.6.5.

2.6.1.8.3. Mating surface with other surfaces

Conditions for mating a surface with other surfaces existing in the model shall be specified during surface creation or edition on the **Mating** tab of the Property bar (Fig.2.6.19).

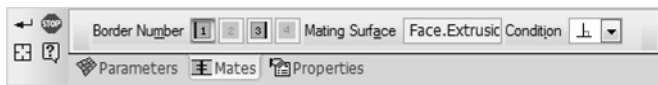


Fig. 2.6.19. **Settings** tab

The created surface can be mated with existing ones along its boarder. In order to perform a mating operation the boarder shall be created as a directly specified curve (i.e. it shall neither be a point nor determined automatically).

Surface boarders available for mating are called **mating boarders**, and in-model surface which is engaged in mating with the create one— **mating surface**. The mating surface shall have a full-length mating boundary. The following may be used as mating surfaces:

- ▼ faces of bodies and surfaces,
- ▼ auxiliary and coordinate planes.

There are three mating conditions available: **Not Specified**, **By Tangent** and **Perpendicularly** (see Section 2.5.1.7).

If the grid contains no curves opposed to the mating boarder (i.e., the curves having common points with the mating boarder), any of the three mating conditions become available for such boarder.

If opposite direction curves exist, all of them shall be perpendicular to the mating surface (**perpendicular** mating), or tangent (in case of mating **by the tangent**). If the opposite direction curves are arranged arbitrary with respect to the mating surface, there is only one mating condition - mating **Not Specified**.

Generally, surface constructed by grid points has 4 boarders, that is why group **Border number** comprises 4 switches (see Fig. 2.6.19). Among them only those are enabled that correspond to the respective mating boarders. Their numbers are displayed in the model window next to the boarders.

To create mating along any mating boarder complete the following steps.

1. In group **Units make** active one of switches. The border will be highlighted in the model window.
2. Select the mating surface in the model window. Name of the guide object will appear in the **Mating Surface** field.
3. Select a mating condition from the **Condition** list. Conditions that can not be applicable to the current boarder are unavailable.

The ghost of the surface being created will move in the model tree in accordance with the selected mating condition; identification of the mating condition will appear near the boarder number.

2.6.1.9. Ruled Surface

A ruled surface is constructed by moving a straight line in space.

KOMPAS-3D allows building a ruled surface along two guide elements one of which may be represented by a point. Some examples of ruled surfaces are given on the figure 2.6.20; the guide curves are marked with a thickened line whereas the guide point is marked with a circle.

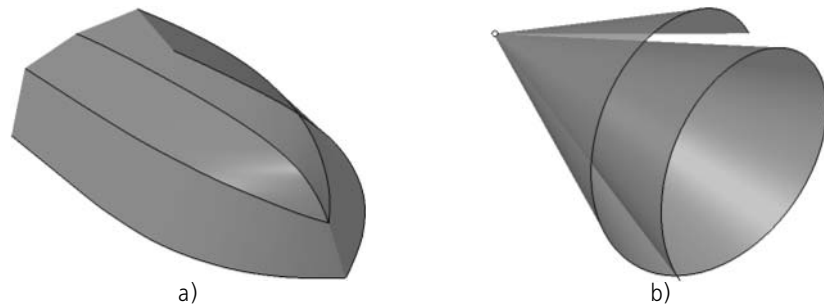


Fig. 2.6.20. Ruled surfaces:
a) along guide curves, b) along a guide curve and a guide point

2.6.1.9.1. Creating a Ruled Surface



To create a ruled surface, click the **Ruled Surface** command. Guideway controls will become available on the Property bar.

Available guideway controls:

- ▼ a 3D curve,
- ▼ a sketch line,
- ▼ an edge,
- ▼ Chain of objects listed above in any combination,
- ▼ a point object.

The objects belonging to 3D curves and point objects are listed in Table 2.1.7 on p. 131.



After calling the command the **Curve 1** switch will be automatically activated.

Specify an object or objects to be used as the first guideway. The thread of 3D curves should be specified in the order of their connection. Name of the selected object or number of objects in the selected succession will appear in field **Curve 1**.



Enable switch **Curve 2** and, following the procedure specified above, select an object or objects to be used as the second guide.

The ghost surface with numbered edges will appear on the screen. The guides are marked with boxed numbers 1 and 2.

To change any of the objects enable the corresponding switch and re-select the object.



To specify a point instead of a curve deselect the curve, and visa versa.



To cancel selection of all the objects click **Re-select** button on the Advanced Control panel.

If you want to check surface for self-crossing, enable the **Check self-crossing** option.

Tab **Split** on the Property bar allows to set split rules surface by faces (see Section 2.6.1.9.2).

Tab **Properties** serves to define the surface name and color.



Selection of guide lines and preset of surface properties finished, click **Create Object** on the Special Control Panel.



The created surface will appear in the model window, and the surface icon — in the Design Tree.

If the **Self-Intersection Check** option was enabled and returned a positive result the surface will be marked in the Design Tree with an exclamation mark in a red circle which indicates errors. For more information about self-intersection of surface refer to Section 2.6.1.6.5.

2.6.1.9.2. Managing Surface Splitting into Faces

By default a ruled surface is split into surfaces automatically - i.e. the number of transversal ribs and the position of their vertices are system-defined.

If one of the guides is a point whereas the other is a single-segment 3D curve, and also in case both the guides are such curves, the surface has two extreme edges — the initial and the final ones. If at least one of the guides is a multi-segment 3D curve or a thread of curves, the surface is split into faces by the edges each of which originates from the guide vertex.

If necessary, the user may change the settings of face splitting into edges. In the example shown on Fig.2.6.21 one automatically generated edge was deleted whereas the vertices of the other two were combined with the guide vertex.

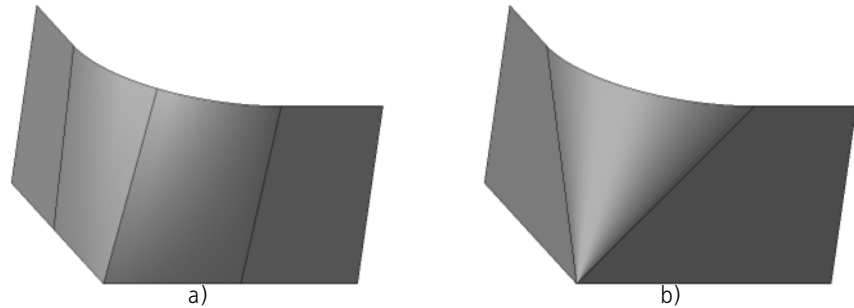


Fig. 2.6.21. Surface splitting into faces: a) automatic, b) user defined

Surface splitting into faces is set up using the **Splitting** tab of the Properties bar.

If option **Auto Determination** is enabled, the surface will be split by faces in automated mode. If the guide has the same number of vertices, you may enable the **By Vertices** option — a surface will be created so that corresponding vertices of guides were connected by edges. With the option disabled the surface with minimum curvature is created. If the number of vertices of guides is different, status of the option does not affect the shape of surface.

To edit the splitting settings disable the **Autodefine** option. Controls for the split process will appear on the **Split** tab (Fig. 2.6.22).

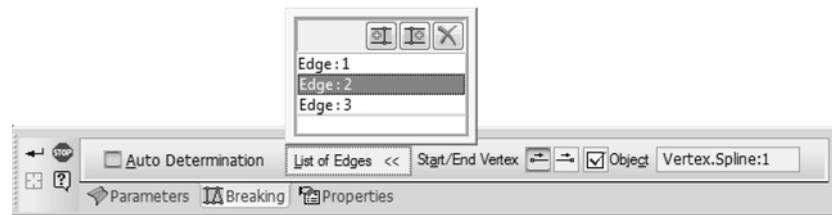


Fig. 2.6.22. **Splitting** tab

Panel **List of Edges** contains numbered list of the surface edges.

When you select a row in the **List of Edges**, the corresponding edge is highlighted in the model window and vice versa, when you select an edge, the row corresponding to this edge is highlighted in the list.

The user may delete and add edges and move edge vertices along the guides, as well as create or cancel the associative relation with the guide vertices as necessary. All the changes will be reflected in the surface ghost.



Absence of the ghost indicates nonexistence of surface. Typical reasons for this are:

- ▼ Matching or intersection of edges;
- ▼ Absence of edges originating from the intermediate vertices of the guides.

Delete Edge



1. Select an edge.
2. Click the **Delete Edge** button on the **List of Edges** bar.
The selected edge disappears while the rest will be re-numbered.
Deletion becomes impossible if the list is reduced to two edges.

Add Edge



1. Select the edge before or after which you want to insert a new edge.
2. Click **Insert Before the Current Edge** or **Insert After the Current Edge**.
A new edge with the successive number is added to the surface.



Associative relation of edge vertices with point objects Relocation of Edge Vertices

By default edge vertices are associated with the guide vertices. The user may cancel the relations and create new ones, if necessary. The relation object may be represented by the guide vertex or a point constructed on it using the **Point** command (the **On Line, Intersection**) or **Group of Points Along a Curve**.

To set the object for mating the edge vertex:



1. Select an edge.
2. Select the edge vertex by enabling the necessary switch in the **Initial/Final Vertex** group. If the vertex is not associated, the handle will be displayed on selecting its edge. Click the handle to select such vertex.
3. Specify the guide vertex or the point on it which the selected edge vertex is to be associated with in the model window. The vertex will be matched with the specified object, and its name will appear in the **Object** field. The “tick” on the **Object** button indicates the relation of the edge vertex with the object.



To cancel the relation of a vertex with an object highlight an edge, select the vertex and click the **Object** button. The tick on the button will disappear.

A non-related vertex can be moved along the guide at will by dragging its handle with the mouse.



If the 3D line used as the guide is hidden, the edge cannot be snapped to the guide vertex.



To use only a segment of the guide instead of its entire length cancel the relation of the extreme edge with the guide vertex and then move the vertex along the guide or relate it to another object.



If one of the guides is represented by a point, the **Initial/Final Vertex** group of switches is not available whereas the relation presence or absence indicator represented by the **Object** element relates to the edge vertex which lies on the guide.

2.6.1.10. Patch

Patch — surface, limited by a closed contour. Edges of bodies and surfaces, sketch lines or 3D curves can be used as contour segments.

Segment specification is subject to the following constraints:

- ▼ The contour shall not have self-intersections.
- ▼ If segments are on the same plane or on a single existing plane, there can be any number of them; in other cases, their number is limited by two and by four.



To create a patch, apply the **Patch** command.

In the Model Tree, specify the contour segments in the sequence they are connected.



If the contour surrounding the patch is contained in one sketch, select that sketch in the Design Tree or in the Model window.

The selected objects will be highlighted in the Model window and in the Design Tree.

The list of objects making up the contour will appear on the Property bar (Fig. 2.6.23). The **Delete** button located above the bar allows deleting some object from the contour. To exclude an object, you can select it again. An excluded face becomes deselected.

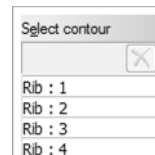


Fig. 2.6.23.

If you want to check surface for self-crossing, enable the **Check self-crossing** option.



To confirm the operation click the **Create Object** button on the Advanced Control panel.



Note that the **Create Object** button is available, if the selected objects make a closed chain.



The surface created shall appear in the model window and the applicable icon — in the Design Tree.

If the **Self-Intersection Check** option was enabled and returned a positive result the surface will be marked in the Design Tree with an exclamation mark in a red circle which indicates errors. For more information about self-intersection of surface refer to Section 2.6.1.6.5 on

p. 454.

2.6.1.11. Blend Surface

You can create a surface that connects two faces. The joint surface boundaries are specified edges or chains of edges of faces to be joined. An edge chain must meet the following criteria:

- ▼ all edges belong to a single face,
- ▼ edges have a common tangent in connection points.

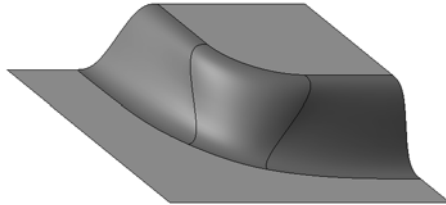


Fig. 2.6.24. Surface, that connects two flat faces

The joint surface is mated with the edges along boundaries. Several mate options are available. You can change the form of a joining surface by means of tension parameter. In addition, you can control the shape of surface lateral edges.

Do the following to create a blend surface.



1. Start the surface creation operation. To do this, click the **Blend Surface** button of the **Surface** panel or call the **Operations — Surface — Blend Surface** command.



2. The system expects indicating the first surface of the surface to be created — in the group **Select boundary**, the **Boundary 1** switch is enabled.

Select the first border in the model window— an edge or chain of edges. Select chain in compliance with edges connection order. Name of the selected edge (or number of edges in a chain) appears in the **Border Curve 1** field. The border is highlighted in the model window, ID number and default mating condition icon are displayed near it.

Set the parameters of the blend surface along its first boarder.

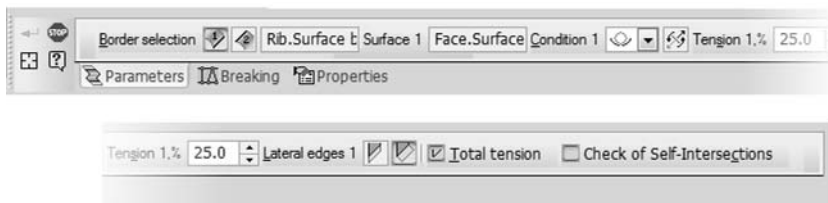


Fig. 2.6.25. Control elements for blend surface parameters of the Properties bar

2.1. Mating surface

As the mating surface, the face with the indicated boundary is selected. The face is highlighted in the model window, its name appears in the **Surface 1** field. If its edges simul-

taneously belongs to two faces, you can change the face. For this, specify the required face in the model window.

2.2. Mating condition and direction

Select the mating condition of the blend surface with the mating surface from the **Condition** list. The following mating conditions are possible: not specified, by tangent, perpendicularly and smooth (see Section 2.5.1.7 on p. 332).



The **Change Direction** switch allows changing mating direction to a reverse one. Meanwhile, the joint surface form near the boundary will change while the mating condition remains the same.

2.3. Tension

This parameter has an effect on a surface form: the more the tension, the more the curvature of the blend surface near the border, and vice versa, the less the tension, the less the curvature. Tension is set as percentage, from 0 to 100.

Enter the tension value in the **Tension, %** field.

2.4. The position of lateral edges

Side edges of the joint surface can be continuations of side edges of surfaces to be joined. They also can be perpendicular to the boundary (Fig. 2.6.26).

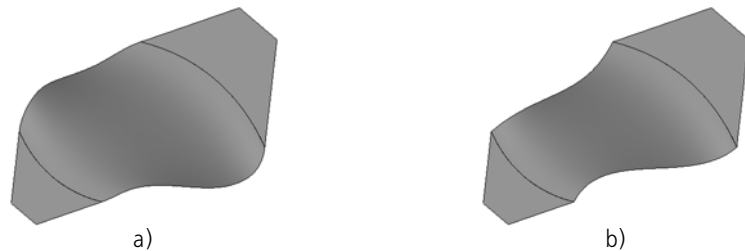


Fig. 2.6.26. Lateral edges of the blend surface:
a) extensions of lateral edges of mated surfaces;
b) are perpendicular to borders



To select a variant of creation of side edges near the first boundary of the joint surface, enable the required switch in group **Side edges**.



3. If you need to use the same tension value for the second border as for the first, enable the **General Tension** option. In this case, when changing tension for one of the boundaries, tension of another is also changed automatically.



4. In the **Boundary selection** group, activate the **Boundary 2** switch. Specify the second border and set the joining parameters along it in the model window— as well as the first border and the joining parameters along it.

The ghost of blend surface with the current parameters appears in the model window. Handles of the ghost allow to change surface tension on the borders.

The second boundary may be of the same face as the first one — the surface will built automatically to connect different edges of one face.

5. If you want to check surface for self-crossing, enable the **Check self-crossing** option.

6. If necessary, you can customize the blend **surface splitting** into faces — move, add or delete surface edges. Enable tab **Splitting** of the Property Bar. Setting up the joint surface splitting is similar to setting up splitting of a linear surface (see Section 2.6.1.9.2 on p. 462) with the only exception: while working with a blend surface the **By Vertices** option is not active.
7. If necessary, change the name and properties of the created surface. Enable the **Properties** tab of the Property Bar.



8. Confirm surface creation. To do this, click the **Create Object** button on the Advanced Control panel.

The created surface will appear in the Model window and its icon will show in the Design Tree. If the **Self-Intersection Check** option was enabled and returned a positive result the surface will be marked in the Design Tree with an exclamation mark in a red circle which indicates errors. For more information about self-intersection of surface refer to Section 2.6.1.6.5 on p. 454.

2.6.1.12. Equidistant Surface

The user may build a curve equidistant to the base one.

The following objects can be used as a base surface.

- ▼ face of the body or surface,
- ▼ Connected set of faces of the same body or surface (see Section 2.5.1.1 on p. 327).

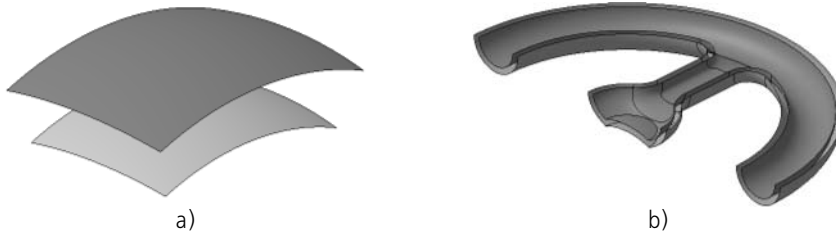


Fig. 2.6.27. Equidistant Surface:
a) Base surface— surface face;
b) Base surface— set of faces



To create an equidistant surface run the **Equidistant to Surface** command.

Specify the base surface in the model window.

If the body/surface faces are used as the base surface, such body/surface can be selected in the Design Tree.



The base surface may be also selected before the command selection.

Second selection of the base surface cancels its selection.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.

The name of the selected object or the number of selected objects are shown in the **Surface** field.



The equidistant surface ghost with the handle for setting the distance from the base surface to the one being created is displayed on the screen (handles are detailed in Section 2.1.4.2.2).



To specify which side of the base surface the equidistant surface will be created enable the **Forward Direction** or **Reverse Direction** switch in the **Direction** group.

Set the distance from the base surface to the one being created by entering it into the **Distance** field on the Properties bar.

All the parameter changes will be displayed on the ghost of the surface being created in the model window.



To change the base surface click the **Re-select** button on the Advanced Control panel and re-select the object or objects.



The **Properties** tab enables setting the name and color of the surface being created.

To finalize construction of the equidistant surface, click the **Create Object** button on the Special Control Panel.



The surface which has been created will appear in the model window and its icon will show in the Design Tree.



At zero shift the equidistant surface copies the base surface. Thus, it is possible to create the required quantity of copies of the surface being base for a few bodies or parts.

2.6.2. Surface editing

2.6.2.1. Surface Trimming

The user may trim a part of the surface or make a cut the shape of the objects lying on the surface or intersecting it (Fig. 2.6.28). Trimming may also follow a contour projected onto the selected surface (see Fig. 2.6.29).

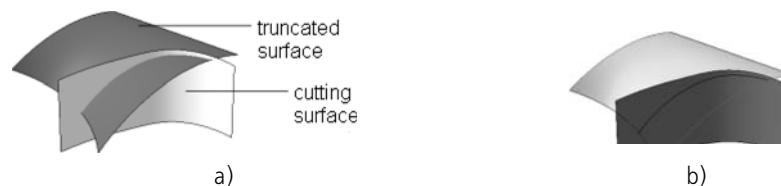


Fig. 2.6.28. Truncate surface using another surface:
a) Surfaces before trimming; b) Trimming result

2.6.2.1.1. Requirements to Objects of the trimming operation

The **trimmed surface** may be represented by a connected set of faces of the same body or surface. When specifying an object in the Design Tree the trimmed surface is formed using all the faces of the object.

The options for **cutting objects** are given in Table 2.6.1. The cutting objects must unambiguously determine the truncation boundary - i.e. completely cross the trimmed surface.



In case of a polyface trimmed surface, the trim boundary is also considered fully defined, if the trimming object intersects one or several faces (not all of it) but in such a way that outermost points of the intersection line are located on the edges of these faces. In this case, edges or parts of surface edges are included in the trimming boundary, see Fig. 2.6.30.

Tab. 2.6.1. Cutting objects while trimming a surface

Cutting Object	Usage Details
Coordinate or Auxiliary Plane	Trimming follows the line of intersecting the plane and the trimmed surface.
Connected set of faces of the same body or same surface	Trimming follows the line of intersecting the cutting object and the trimmed surface. When specifying an object in the Design Tree the cutting object is formed using all the faces of the selected object.
Curve on Trimmed Surface:	Trimming follows a curve.
▼ a 3D curve,	A 3D line may represent the cutting object only if this curve is directly initial or directly derivative * in relation to the trimmed surface.
▼ solid or surface edge,	
▼ Chain of curves or edges.	<ul style="list-style-type: none"> ▼ Example of using a source curve: cutting a surface built along a curve grid (see Section 2.6.1.8 on p. 458), by one of the curves. ▼ Example of using a derivative curve: cutting the surface along the equidistant curve built by offset along the trimmed surface (equidistant curves are covered in Section 2.5.4.17 on p. 417).
Sketch	Trimming follows the line of intersecting the trimmed surface and the surface produced by extruding a sketch perpendicularly to its plane (see Fig. 2.6.29). The sketch must be specified in the Design Tree. A sketch may contain one open contour or multiple closed contours. The contours must not self-intersect.

* In other words the hierarchy of the relations between the curve and the trimmed surface must not contain any other objects (viewing object relations is covered in Section 2.1.2.7.2 on p. 114).

Model object selection is detailed in Section 2.1.4.1.1 on p. 130.

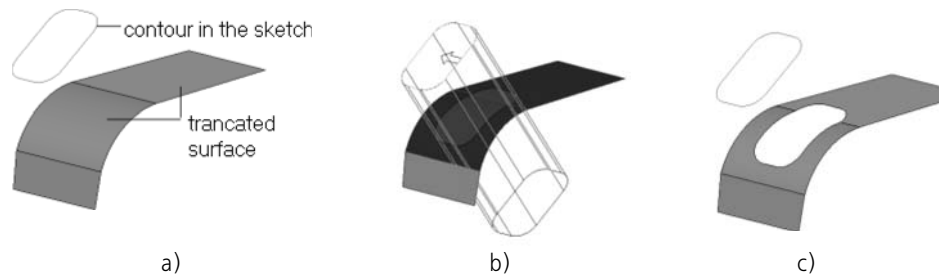


Fig. 2.6.29. Example - Truncation of surface using the sketch contour:
a) surface to be truncated and secant object, b) objects after selection, c) truncation result

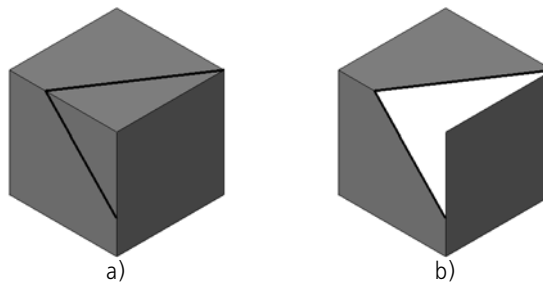


Fig. 2.6.30. Example of triangular surface trimming:
a) surface to be truncated and secant object, b) truncation result

2.6.2.1.2. Trimming Operation



To trim a surface run the **Surface Trimming** command. The command is accessible if there is at least one body or surface in the Model.

The trimming controls will appear on the Properties bar (Fig. 2.6.31).

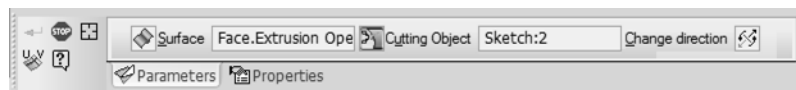


Fig. 2.6.31. Properties bar while trimming a surface



The **Surface** switch is enabled automatically. Specify the faces forming the trimmed surface in the model window. The surface object or body can be selected in the Design Tree.



Surface to be truncated can be also selected before calling the command.



To specify the object the intersection with which defines the line of plane truncation, enable the **Secant Object** switch.

Specify the secant object(s) in the model window or in the Design Tree.



If it is required to trim a surface along the isoparametric curve, this curve may be built without leaving the trimming process. To do this, click the **Isoparametric Curve** button on the Advanced Control panel. The isoparametric curve on the surface creation operation will be started (see Section 2.5.4.18 on p. 422). If the trimmed surface consists of one surface, it is selected automatically. In case of several surfaces, you should specify the face on which the curve should be positioned. After that, specify the point on face, choose the direction of isoparametric curve passing through it, and click the **Create Object** button. The system will resume the trimming operation process, created curve will appear in the Design Tree and will be automatically selected as a secant object.

Trimmed surface and secant object are highlighted and their names appear in the **Surface** and **Secant Object** fields.

The arrow pointing to the surface part to be cut is displayed at the object intersection line.



The **Change Direction** switch allows to select another part of the trimmed surface to be truncated.

To deselect an object as an object to be truncated or a secant object, enable the **Surface** or **Secant Object** switch respectively, and reselect an object in the Model window. Highlighting will be cleared off the object.



To cancel selection of all the objects click the **Re-select** button on the Advanced Control panel and re-select objects.

The name of the trimming operation can be edited in the **Properties** tab of the Property bar.



To complete surface trimming, click the **Create Object** button on the Special Control Panel.



The **Create Object** button will become unavailable if the selected objects fail to meet the criteria (see Section 2.6.2.1.1 on p. 469).



The truncated surface will be displayed in the model window and the corresponding icon — in the Design Tree.

If the user has specified a body or its faces as the trimmed object, the trimming operation may cause disintegration of the body. For details on bodies with broken integrity, see Section 2.6.2.6 on p. 480.

2.6.2.2. Surface splitting

You can split a surface or body faces into several new faces. With this, the model gets additional edges representing the boundaries of the newly generated facets (Fig. 2.6.32). Surface splitting can be done:

- ▼ by curves on surface,
- ▼ by the lines of intersection with other surfaces or planes,
- ▼ by contours, projected to the surface.



Fig. 2.6.32. Using an edge resulting from face's splitting:
a) initial state of a part, b) result of a rounding by maintaining "cliff" edges

2.6.2.2.1. Requirements to Objects of the splitting operation

As a **surface to be trimmed**, you can use any set of faces, bodies, and/or surfaces. When specifying an object in the Design Tree the split surface is formed using all the faces of the object.

The options for **cutting objects** are given in Table 2.6.2. The cutting objects must unambiguously determine the split line - i.e. completely cross the split surface.



In case of a polyface trimmed surface, the splitting line is also considered fully defined, if the trimming object intersects one or several faces (not all of it) but in such a way that outermost points of the intersection line are located on the edges of these faces. In this case, the operation will result only these faces splitting.

Tab. 2.6.2. Cutting objects while trimming a surface

Cutting Object	Usage Details
Coordinate or Auxiliary Plane; Face	<ul style="list-style-type: none"> ▼ Trimming follows the line of intersecting the plane or face and the splitted surface. ▼ You can use any set of planes and/or faces.
Curve on Split Surface:	<ul style="list-style-type: none"> ▼ Trimming follows a curve.
<ul style="list-style-type: none"> ▼ projection, ▼ isoparametric, ▼ intersection line, ▼ outline line, ▼ spline on surface 	<ul style="list-style-type: none"> ▼ You can use arbitrary set of curves. ▼ Curves can intersect but must not have self-intersect.

Tab. 2.6.2. Cutting objects while trimming a surface

Cutting Object	Usage Details
Sketch	<ul style="list-style-type: none"> ▼ Trimming follows the line of intersecting the trimmed surface and the surface produced by extruding a sketch perpendicularly to its plane. ▼ You may use only one sketch. ▼ Sketch objects can be a part of arbitrary number of closed and/or open contours. Contours can intersect but must not have self-intersect.

Model object selection is detailed in Section 2.1.4.1.1 on p. 130.

2.6.2.2.2. Splitting performing



To split a face (faces) of a surface or a solid, apply the **Split surface** command. The command is accessible if there is at least one body or surface in the Model.

The splitting controls will appear on the Properties bar (Fig.2.6.33).

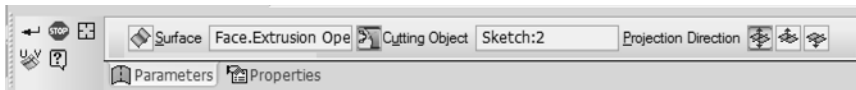


Fig. 2.6.33. Properties bar while splitting a surface



The **Surface** switch is enabled automatically. Specify the faces forming the splitted surface in the model window. The surface object or body can be selected in the Design Tree.



The surface to be split may be also selected before the command selection.



To specify the object the intersection with which defines the line of plane splitting, enable the **Secant Object** switch.

Specify the secant object(s) in the model window or in the Design Tree.



If it is required to split a surface along the isoparametric curve, this curve may be built without leaving the trimming process. To do this, click the **Isoparametric Curve** button on the Advanced Control panel. The isoparametric curve on the surface creation operation will be started (see Section 2.5.4.18 on p. 422). If the splitted surface consists of one surface, it is selected automatically. In case of several surfaces, you should specify the face on which the curve should be positioned. After that, specify the point on face, choose the direction of isoparametric curve passing through it, and click the **Create Object** button. The system will resume the splitting operation process, created curve will appear in the Design Tree and will be automatically selected as a secant object.

Split surface and secant objects are highlighted and their names appear in the **Surface** and **Secant Object** fields. The ghosts of edges to be formed during splitting will appear in the model window.



If a sketch is selected as the secant object, the **Projection Direction** group of switches appear on the Properties bar and allow to set up the direction, in which the sketch will be moved to generate the secant surface. Activate the desired switch: **Both Directions**, **Forward Direction** or **Reverse Direction**. If direct or reverse movement direction is selected for the sketch, the direction is shown by an arrow on a ghost in the model window.



All contours of the specified sketch take part in the trimming operation. You cannot use separate contours.

To deselect an object as an object to be split or a secant object, enable the **Surface** or **Secant Object** switch respectively, and reselect an object in the Model window. Highlighting will be cleared off the object.



To cancel selection of all the objects click the **Re-select** button on the Advanced Control panel and re-select objects.

The name of the splitting operation can be edited in the **Properties** tab of the Property bar.



To complete surface splitting, click the **Create Object** button on the Special Control Panel. Selected faces are split into several faces, and in the Design Tree the surface splitting operation icon appears.

2.6.2.3. Surface Extension

The edge of a surface or the face of a body suffered from disintegration beyond the boundaries of the specified edges of the face — open edges - may be extended (open edges are detailed in Section 2.6.2.4 on p. 478).

The extension operation changes positions of or sets new faces adjacent to the specified borders.

A face is extended over the specified borders for the preset distance or up to the specified point object.

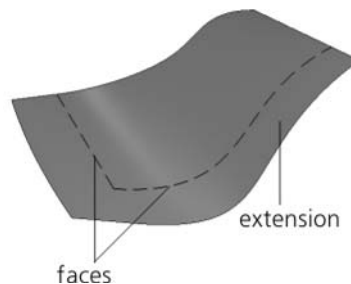


Fig. 2.6.34. Face Extension



To extend a face run the **Surface Extension**.

Select a face boarder(s) in the Model window. If you need to extend a face over all boarders, select this face.



The face or its borders can be selected in the Model window prior to calling the respective command.

The extension ghost with a handle for setting the face extension value will appear on the screen (handles are detailed in Section 2.1.4.2.2).

To cancel object selection, re-select it.

The object selection is detailed in Section 2.1.4.1.1 on p. 130.



The name of the selected object or the number of selected objects are displayed in the field next to the **Edges** switch on the Properties bar.

The **Extension Type** group of switches enables selection of the face extension type: by the same surface, tangentially or in the direction (see Section 2.6.2.3.1 on p. 476).

The **Method** drop-down list enables selection of the face extension method (see Section 2.6.2.3.2 on p. 478).

All changes are displayed in the extension ghost in the model window.



To change objects used for the face extension, click **Reselect** button on the Special Control Panel and re-select objects.



Controls in the **Property** tab of the Property bar set the object' name and color.

To finalize face extension click the **Create Object** button on the Advanced Control panel.



The face extension will appear in the model window and its icon will show in the Design Tree.


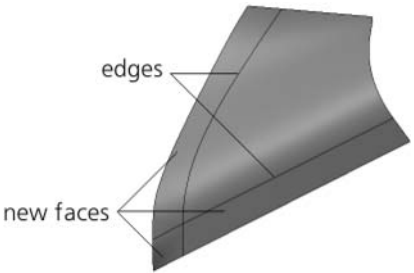

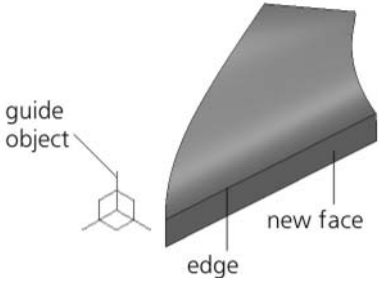
2.6.2.3.1. Extension Type

To select the face extension type, enable the desired switch in the **Type** group (Table 2.6.3).

Tab. 2.6.3. Extension Types

Extension Type	Construction result	
	<p>By the same surface</p> <p>This face extension type is set by default. In this case the face is extended without adding any new faces — the face is extended beyond the open edges along its theoretical surface (theoretical surface is detailed in Section 2.5.1.3 on p. 328). The user may choose the position of the lateral extension edges (see below).</p>	

Tab. 2.6.3. Extension Types

	Extension Type	Construction result
	<p>By Tangent Extension is performed by creating new faces tangential to the source face along the designated edges. The user may choose the position of the lateral extension edges (see below).</p>	
	<p>In the Direction Extension is performed by creating new faces formed by moving the designated edges in the specified direction. The direction is set by the guide object or vector (see below).</p>	

Position of Lateral Extension Edges

If the user has chosen the **By Same Surface** or **Tangentially** face extension method, the following options of its side edges are available: extension of source side edges or extension along the normal towards selected boards.

The various position of lateral edges are shown on Fig. 2.6.35. The dotted line corresponds to the edge beyond which the face is extended.

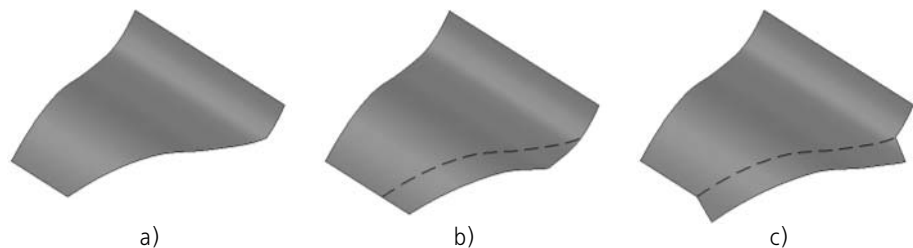


Fig. 2.6.35. Various positions of lateral edges: a) source object, b) extension of the original lateral edges, b) extension along the normal toward the specified edges

To select the position of lateral edges use the **Lateral Edges** group of switches. Activate the desired switch:



▼ **Extension of the Source Side Edges;**



▼ **Along the Normal to Selected Boards.**

Specifying Extension Direction

Any straight or flat object can be used as the guide object to extend a face. Straight and flat objects are listed in Table 2.1.7 on p. 131.

Direction, defined by a straight object — line, parallel to the object.

Direction, defined by a plane object — line, perpendicular to the object.



Moreover, the guide object may be represented by a vector. Such vector can be created by means of **Vector drawing** command (see Section 2.8.5 on p. 613).



To preset the face offset direction specify a guide in the Design Tree or construct a vector.

The name of the selected object shall be displayed in the field beside the **Direction** switch. If the direction is set by means of a vector, the field will contain the word “Vector”.

To change the guide object, enable switch **Direction** and reselect a new guide object or construct a vector.



The **Change Direction** switch enables reversing the edge extension direction.

2.6.2.3.2. Extension Method



Two edge extension methods are available:

- ▼ **To Specified Length** — the edge is extended beyond the specified boundaries to the specified distance;



- ▼ **Toward Vertex** — the edge is extended beyond the specified boundaries toward the selected point object.

Select the extension method using the **Direction** list on the Property bar.

- ▼ If the **To Specified Length** method is selected, the **Length** field will appear on the Properties bar.

Enter the desired extension value in the field.

- ▼ If the **Toward Vertex** method is selected, the **Vertex** control will appear on the Properties bar.

Specify the desired point object in the Model window (point objects are listed in Table 2.1.7 on p. 131). The name of the selected object shall be displayed in the field beside the switch.



To change the point object enable the **Vertex** switch and specify the desired object in the model window.

2.6.2.4. Surface Knitting

KOMPAS-3D enables joining of open edges of a single or multiple surfaces (Fig. 2.6.36 a, b) resulting in the formation of a whole surface and attachment of the surface (surfaces) to the open edges of the body (Fig. 2.6.36, c) with broken integrity. For details on bodies with broken integrity, see Section 2.6.2.6 on p. 480.

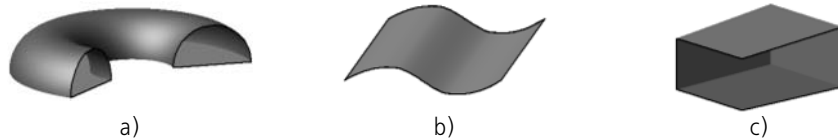


Fig. 2.6.36. Examples of open edges (marked with black color): a) revolution surface, b) extrusion surface, c) a body with broken integrity



To join the surfaces, apply the **Surface Knitting** command.

In the model window select the objects to be knitted. Those will be highlighted with a color. The list of knitted objects will appear on the Property bar. Using the buttons located above the list, you can change the order of surfaces and delete them from the list.

To exclude any surface from the pattern to be knitted re-select it in the model window.

In field **Accuracy** on Property Bar you should specify a maximum gap between the edges of the surfaces to be knitted.

The **Create Solid** option allows creating a body surrounded by the knitted surfaces. If a parts with broken integrity is present in the list of knitted surfaces, then the **Create Solid** option turns On automatically and cannot be disabled.



To confirm surface knitting, click the **Create Object** button on the Special Control Panel. If the list of surfaces to be knitted contains only one object, the **Create Object** button will only be accessible with the **Create Body** option enabled.



Surfaces, whose neighboring edges' distances are less or equal to the preset accuracy value, will be integrated into a single surface. In the Design Tree, a knitting icon appears.

Once solid construction has been enabled, then the created surface shall be checked for closure. In case of success, the resulting solid body will be limited by the obtained surface, and in case of failure, the resulting body will have broken integrity.

2.6.2.5. Deletion of Faces



To delete a face (faces) of a surface or a solid, apply the **Delete Faces** command.

Select faces to be deleted in the Model window. Those will be highlighted with a color.

Number of deleted faces shall appear in similar field on tab **Parameters** of Property Bar.

To exclude some face from selected for deletion, click it in the part window again. The face will be deselected.

To confirm splitting selected faces, click the **Create Object** button on the Special Control Panel.

Selected faces will be deleted from the model.



In the Design Tree, the face deletion icon will appear.



When deleting faces of bodies, the model is marked as an error in the Tree. The reason for that is broken integrity of bodies (see Section 2.6.2.6).

2.6.2.6. Bodies with Broken Integrity

The body integrity is broken if it acquires any open edges similar to the open edges of open surfaces (see Fig. 2.6.36 on p. 479).

The following may lead to appearance of bodies with broken integrity in the model:

- ▼ Deleting body edges using the **Delete Edges** command (see Section 2.6.2.5);
- ▼ Deleting a part of the body surface using the **Surface Trimming** command (see Section 2.6.2.1 on p. 469);
- ▼ surface knitting using the **Surface Knitting** command (see Section 2.6.2.4 on p. 478), if the **Create Body** has been enabled during the operation but the connected faces did not form a closed surface.



A special icon appearing in the Design Tree instead of the whole body icon indicates loss of integrity. The model is marked as an error in the Model Tree.

Bodies with broken integrity can be edited by means of operations with surfaces — such as extension, trimming etc.

To restore integrity add surface to close the body and close them using the **Surface Knitting** command.



Please note that closing requires mating an open edge of one surface with an open edge of another surface.



Once the integrity is restored, the icon in the Design Tree will be replaced with the icon of the solid body.



The error mark in the model is displayed until the integrity of all the bodies is restored.

2.6.3. Spline form. Face customization

You can edit a selected part's face by transforming it into a spline surface. To do so, use the special customization mode which makes it possible to customize the form and the position of the selected face. In Fig. 2.6.37, you can see an example of the face's form changing in the special customization mode.



Fig. 2.6.37. Face customization:
a) face's original state, b) customization result



In order to enter the face customization mode, call the **Spline form** command.

The compact bar's appearance will change. It will contain buttons for switching between the two toolbars: **Spline form** (enabled by default) and **Filters**. The **Spline form** button is enabled on the **Spline form** toolbar, while the Property bar contains buttons of the Special Control Panel.

Select with the cursor the part's face that needs to be edited.



The edge can also be specified before calling the **Spline form** command.

The specified face will be transformed into a spline surface¹ and highlighted (for more information about the spline surface refer to Section 2.5.1.4 on p. 329). Ghost grid created on the basis of the parametric area of the surface (see Section 2.5.1.3 on p. 328) will be displayed on the screen. The grid is formed by rows of points – poles. The rows of the grid are located in two directions — U and V. When changing the poles positions, the form and/or the position of the edge may change.

If necessary, you can edit the equidistant surface of the selected face (not the face itself). See Section 2.6.3.5 on p. 505.



Note that when editing an existing face, boundary forms of its adjacent faces may change, while the equidistant surface is a separate object. Therefore, its customization does not change the boundaries of the part's faces.

The following actions can also be performed in the face editing mode:

- ▼ changing the form and the position of a face by moving poles using the toolbar commands (see Section 2.6.3.1 on p. 482) or by the mouse (see Section 2.6.3.4 on p. 503)
- ▼ fixation of grid poles and boundaries of the customized face (see Section 2.6.3.2 on p. 499),
- ▼ changing spline form parameters (see Section 2.6.3.3 on p. 501).

1. As a result of the transformation, the face will lose its initial properties. For example, after transformation of a conical face into a spline face, construction of its axis using the **Conical surface's axis** command will become impossible.

You can cancel performed operations or redo the last canceled operation (see Section 1.4.1.6 on p. 79).



Besides that, you can redefine the object for editing by selecting another edge for editing using the **Reselect** button of the Special Control Panel. Note that all modifications of the edge and its equidistant, made before object reselecting, will be lost.



In order to complete face editing, click the **Create object** button. Working in the edge editing mode will be finished, and all changes will be saved. An icon appropriate for the **Spline form operation** will appear in the Design Tree.



The **Interrupt command** button makes it possible to leave the editing mode without saving the changes.

2.6.3.1. Changing poles positions



To change grid poles positions, the following commands of the **Spline form** toolbar are used:



▼ **Linear displacement,**



▼ **Displacement with smoothing,**



▼ **Rotation,**



▼ **Scaling,**



▼ **Alignment.**

You can use these commands to move individual poles or all poles in the selected row. Details about the selection of poles can be found in Section 2.6.3.1.1 on p. 483.

When changing the poles positions, the form and/or the position of the face may change. If the new position of poles does not ensure correct changing of the face's form or position, the face remains in its initial state (refer to Section 2.6.3.6 on p. 507 for more details).

Depending on the command, poles' displacement is defined by different parameters: offset value, rotation angle, scaling coefficient and so on. However, each of the above-mentioned commands requires displacement direction setting. Typical displacement direction variants are described in Table 2.6.4. Special variants that are used with certain commands are described in their respective sections.

Tab. 2.6.4. Pole displacement direction variants

Variant	Description
Along the X axis	Direction is defined by X, Y, or Z axis of the current coordinate system (see Section 2.8.3.3 on p. 597).
Along the Y axis	
Along the Z axis	

Tab. 2.6.4. Pole displacement direction variants

Variant	Description
Parallel to Object	<p>Direction is set by specification of an object of one of the following types: straight-line object (see Table 2.1.7 on p. 131), cylinder, cone, toroid.</p> <p>In case of a straight-line object selection, the direction will coincide with it.</p> <p>In case of a cylinder, cone or toroid selection, the direction will coincide with its axis.</p> <p>Specify the required object in the Design Tree or Model Window. The name of the object is displayed in the Guiding Object field.</p> <p>In case of poles movement by rotation, scaling or alignment, the Guiding object switch will be present on the Property bar. Use it to change the guiding object, if necessary.</p>
Perpendicular to Object	<p>Direction is defined by specifying a custom flat object (see Table 2.1.7 on p. 131) and coincides with the perpendicular to the object. After selection of this object, select the desired object in the Design Tree or in the model window. The name of the object is displayed in the Guiding Object field.</p> <p>In case of poles movement by rotation, scaling or alignment, the Guiding object switch will be present on the Property bar. Use it to change the guiding object, if necessary.</p>
By Normal	<p>The direction is set by the normal to the edited edge going through the pole which is being displaced.</p>

2.6.3.1.1. Selection of poles

Depending on which command is used for displacement or fixation of poles, the poles for displacement are selected as follows.

- ▼ In order to select poles for displacement by **Rotation, Scaling, Alignment** commands, as well as for fixation using the **Fix poles** command, do the following.



Move the cursor to the required pole. As soon as the symbol of a vertex appears next to the cursor, click the mouse. The selected pole will be highlighted. Select all the required poles one by one. If a pole's selection should be canceled click it again with the mouse.

You can select individual poles and rows of poles. Selection of a row means that all poles of this row are added to the set of selected poles.



To select a row of poles, move the cursor to the segment of the row. As soon as the symbol of a 3D curve appears next to the cursor, click the mouse. All segments of the selected row will be highlighted. The repeated mouse button pressing on the row's segment cancels the selection of poles.

- ▼ In order to select poles for displacement by the **Linear displacement** or **Displacement with smoothening** command, poles or segments of pole rows should be specified by clicking the

mouse keeping the **<Ctrl>** key pressed. Another mouse click with pressed **<Ctrl>** key deselects the pole (pole row).



Poles of the selected row can also be selected individually (before or after the row selection). These poles remain selected after row's deselection.

You can select any number of poles by specifying separate poles or grid rows.



You can also use **Select all poles**, **Cancel pole selection** and **Invert pole selection** commands from the context menu to select poles.

2.6.3.1.2. Linear pole displacement

Positions of grid poles of the edited face can also be changed by linear displacement (Fig. 2.6.38).

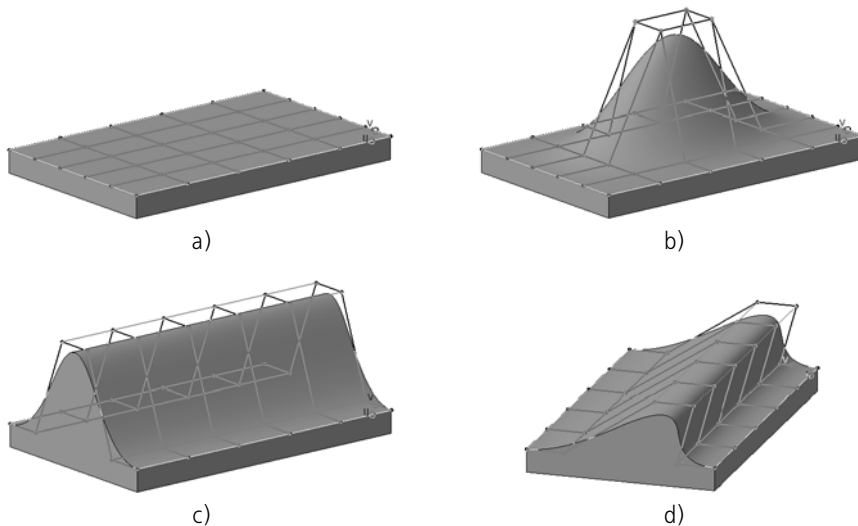


Fig. 2.6.38. Displacement of poles:
 a) face's initial state, b) displacement of poles along the Z axis,
 c) displacement of pole rows along the Z axis, d) displacement of pole rows within the screen's plane

The selected rows are moved in the required direction in accordance with certain displacement conditions.

To move objects, perform the following operations.



1. Call the **Linear displacement** command. The Property bar will display control elements for displacement parameters' settings (Fig. 2.6.39).

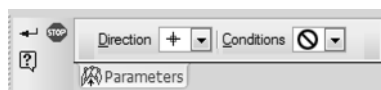


Fig. 2.6.39. Property bar during the displacement of poles

2. Specify poles that should be displaced (see Section 2.6.3.1.1 on p. 483).



Linear displacement of poles can be made manually or automatically.

If you need to move one pole manually (one row of poles), you do not have to specify a pole (row of poles). The displacement procedure is described in p. 5 of the current Section.

3. Select the required pole displacement direction option using the **Direction** drop-down list. The following methods are available:

- ▼ **Along the X axis,**
- ▼ **Along the Y axis,**
- ▼ **Along the Z axis,**
- ▼ **Parallel to an Object,**
- ▼ **Perpendicular to object,**
- ▼ **By normal,**
- ▼ **Along polygon,**
- ▼ **Within the screen plane.**

Description of typical displacement direction variants (**Along the X axis, Along the Y axis, Along the Z axis, Parallel to object, Perpendicular to object, By normal**) can be found in Table 2.6.4 on p. 482.

If the **Along polygon** option is selected, displacement is performed along U or V grid direction.

The **Within the screen plane** option makes it possible to move poles within the screen plane. Additional controls will become available on the Property bar after selection of the displacement direction variant. The set of these controls and further actions depend on the selected variant.

4. Make additional displacement settings in accordance with the selected variant.

- ▼ **Along the X axis/Along the Y axis/Along the Z axis** — no additional settings are required.
- ▼ **Parallel to object/Perpendicular to object** — specify the guiding object in the model Tree or window. The name of the object is displayed in the **Guiding Object** field.
- ▼ **By normal** — enable/disable the **From source** option.

If the option is enabled, the normal's position will be calculated with respect to the edited edge's source form; if the option is disabled — with respect to the current edge's form. The edge's source form is the form that the edge had at the moment of calling the **Spline form** command.



- ▼ **Along polygon** — select the direction type by enabling the **Along U** or **Along V** of the **Type** group of switches. Two offset variants are available for each direction.

- ▼ **To the next,**
- ▼ **To the previous.**

If the first option is selected, the pole will be moved along the grid's segment connecting the pole with the next pole along the U/V direction; if the second option is selected, it will be moved opposite to the U/V direction.

Activate the switch appropriate to the required variant.

▼ **Within the screen plane** — no additional actions are required.

5. Move the selected poles manually or automatically.



To move the poles manually, move the cursor to one of the selected poles. When the symbol of a vertex appears next to the cursor, move the mouse to the required position keeping the left button clicked. After that, release the button.



If selected poles include the poles that were selected by row specification, you can move the cursor to this row's segment. When the symbol of a 3D curve appears next to the cursor, move the mouse to the required position keeping the left button clicked. After that, release the button.



It is impossible to move the poles behind the row's segment, if the **By normal** movement direction is set.

To automatically move the poles, enter the offset value for the poles in the **Offset** field. The value can be both positive and negative. Once the poles have been moved, the **Offset** field will be set to zero.



For the **In the screen's plane** direction of movement, you can only use manual movement of poles.

6. When moving the selected poles manually, you can set an additional condition of the movement making it possible to move some poles of the grid adjacent to the selected one. To do this, use the **Conditions** list. The procedure of setting conditions is described in detail below.



7. To finish the command, click the **Create object** button.

Additional condition of pole displacement

In case of a manual pole displacement, you can move not only the selected poles but also a certain number of nonselected poles located in the same rows with the selected poles. The rows can be located along U and V direction of the grid, as well as in both directions (see Fig. 2.6.40).

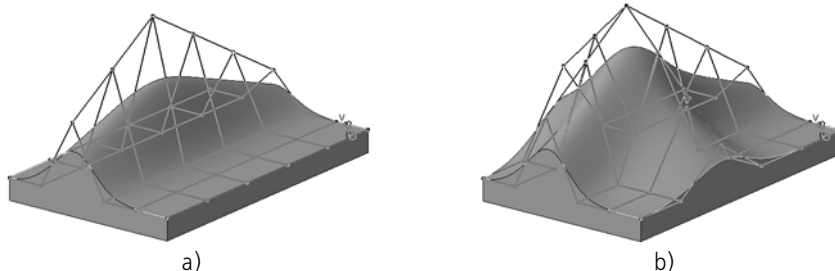


Fig. 2.6.40. Proportional displacement of poles:
a) in the row along the U direction, b) in rows along U and V directions

Poles are moved proportionally along the same direction with the selected poles.

The offset of each poles is calculated as follows.

Let P_0 is a pole selected for manual displacement. Its offset L_0 is defined by the user.

P_i is a pole which is moved together with the selected pole,

where i is a serial number of the pole in the row. The selected pole is the starting point of pole counting. The pole which is closest to the selected one has serial number 1, the following pole has serial number 2, and so on.

The number of poles which are proportionally moved together with the selected one is set by the user and is equal to n , i.e. the first serial pole for displacement is P_1 while the last one is P_n .

Offset L_i of the P_i pole is calculated automatically. The calculation is made in accordance with the formula:

$$L_i = L_0 \cdot (n + 1 - i) / (n + 1)$$

Thus, for example, in case of a proportional displacement of 3 poles (Fig. 2.6.41), the offset of each of them will be as follows:

$$L_1 = L_0 \cdot (3 + 1 - 1) / (3 + 1) = L_0 \cdot 3 / 4,$$

$$L_2 = L_0 \cdot (3 + 1 - 2) / (3 + 1) = L_0 \cdot 1 / 2,$$

$$L_3 = L_0 \cdot (3 + 1 - 3) / (3 + 1) = L_0 \cdot 1 / 4.$$

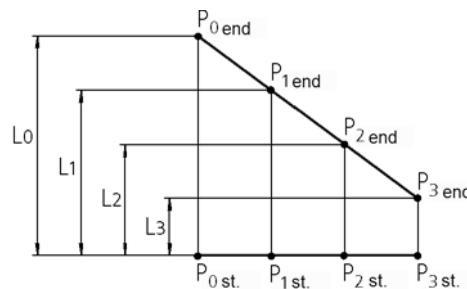


Fig. 2.6.41. Proportional displacement of poles



The specified number n of poles for displacement can exceed the number of poles in a row. In this case, only existing poles will be displaced but the specified number n will be used in the calculation of each pole's offset.

To set up proportional displacement of poles, do the following.

1. In the **Condition** drop-down list, select the **Proportionally** option. Additional controls will become available on the Property bar (Fig. 2.6.42).

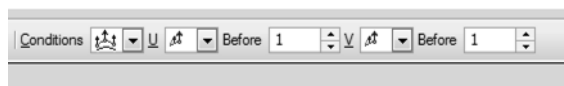


Fig. 2.6.42. Setup elements of proportional pole displacement

2. Set the direction of proportionally displaced pole counting.

The number of proportionally displaced poles is counted in a certain direction starting from the selected pole. The direction of counting in a row is set separately for each direction of the grid (U and V) using drop-down lists **U** and **V**. The following methods are available:

- ▼ **Before and after** — the number of poles to be moved is reckoned in each direction from the selected pole,
- ▼ **Before** — the number of poles to be moved is reckoned from the selected pole against the U/ V direction,
- ▼ **After** — the number of poles to be moved is reckoned from the selected pole in the U/V direction,
- ▼ **No displacement** — only selected pole is displaced; positions of other poles in the row will not be changed.

Select the desired item from each of the drop-down lists.

3. Enter the number of displaced poles in the **Before** and **After** fields for each grid direction separately (U and V). Displayed fields depend on the selected pole counting variant.



If several poles are selected, the preset number is counted in the row for each of them.

4. Move the selected poles manually. Proportional displacement of the defined number of poles will be performed automatically.

2.6.3.1.3. Displacement of poles with smoothening

Positions of grid poles of the edited face can also be changed with smoothening (Fig. 2.6.43).

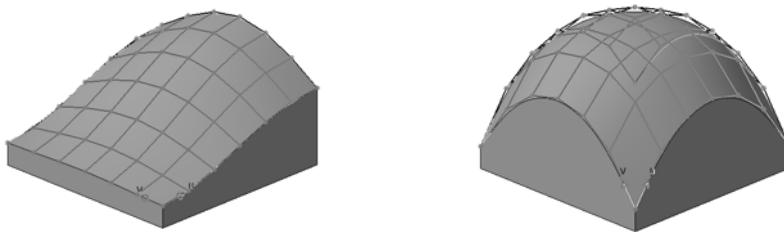


Fig. 2.6.43. Examples of pole displacement with smoothening

The following types of smoothening are available (Fig. 2.6.44):

- ▼ **Smooth transition,**
- ▼ **Convex,**
- ▼ **Concave.**

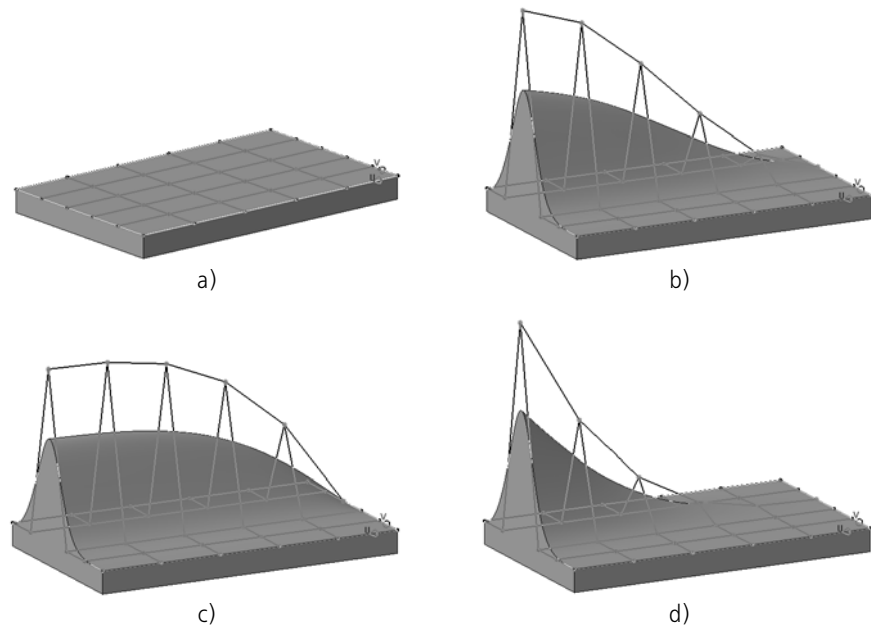


Fig. 2.6.44. Displacement of one row's poles with smoothing:
 a) initial face's state, b) **Smooth transition** type of smoothing,
 c) **Convex** type of smoothing, d) **Concave** type of smoothing

The selected poles are moved in the user-defined direction. In order to move poles, one of them should be displaced by the mouse. The remaining poles are automatically moved in the same direction.

The offset of each poles is calculated as follows.

Let P_0 is a pole which is moved by the mouse. Its offset L_0 is defined by the user.

P_i is a custom pole, one of the selected poles,

where i is the pole's number.

Offset L_i of the P_i pole is calculated automatically using the following formula:

$$L_i = k_i \cdot L_0,$$

where k_i is the smoothing factor for the P_i pole.

In order to calculate the smoothing factor, the following formulas are used:

$k_i = 1 - 3 \cdot x_i^2 + 2 \cdot x_i^3$ — spline function, smoothing of the **Smooth transition** type,

$k_i = 1 - x_i^n$ — exponential function, smoothing of the **Convex** type,

$k_i = (1 - x_i)^n$ — exponential function, smoothing of the **Concave** type,

where n is a natural number; value range of argument x_i $[0;1]$.

Therefore, the formula for calculation of L_i offset of the P_i pole, depending on the selected smoothing type, will look as follows:

$$L_i = (1 - 3 \cdot x_i^2 + 2 \cdot x_i^3) \cdot L_0, \text{ or}$$

$$L_i = (1 - x_i^n) \cdot L_0, \text{ or}$$

$$L_i = (1 - x_i)^n \cdot L_0.$$

The x_i argument is the distance between the pole for which the calculation is made and the straight line going in the direction of displacement through the P_0 pole.

Since the value range of the argument is $x_i [0;1]$, the distance to the outermost pole P_{max} is regarded as a unit segment $x_{max} = 1$. At the same time, the distances to all other poles (located closer) accept values between 0 and 1.

As a result of calculations for the outermost pole P_{max} , the displacement is equal to zero, i.e. this pole does not move. All other poles' displacement values are other than zero.



If, among the selected poles, there are several poles located at the same distance from the straight line going in the displacement direction through the P_0 pole, and this is a minor distance, non of the poles will be displaced.

An example of displacement with the **Smooth transition** smoothing type for 6 selected poles is presented in Fig. 2.6.45.

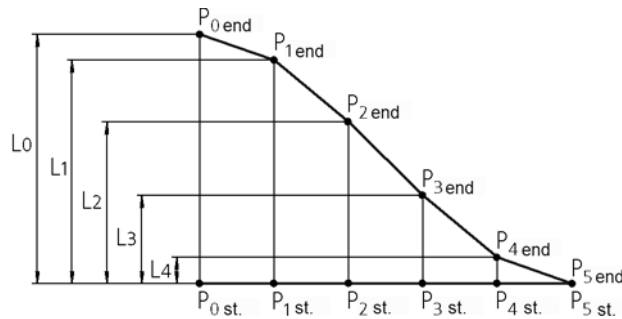


Fig. 2.6.45. Displacement of poles with smoothing of the **Smooth transition** type. The first of the selected poles P_0 is moved by the mouse. The last of the selected poles P_5 is the most distant from the straight line going in the direction of movement through the P_0 pole. The displacement of the P_5 pole is equal to zero.

Displacement with smoothing



1. Call the **Displacement with smoothing** command. The Property bar will display control elements for displacement parameters' settings (Fig. 2.6.46).

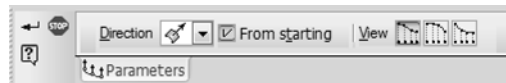


Fig. 2.6.46. Property bar during the displacement of poles with smoothing.

2. Specify poles that should be displaced (see Section 2.6.3.1.1 on p. 483).



If you need to move one pole (one row of poles), you do not have to specify a pole (row of poles). The displacement procedure is described in p. 7 of the current Section.

3. Select the required pole displacement direction option using the **Direction** drop-down list. The following methods are available:

- ▼ **Along the X axis,**
- ▼ **Along the Y axis,**
- ▼ **Along the Z axis,**
- ▼ **Parallel to an Object,**
- ▼ **Perpendicular to object,**
- ▼ **By normal.**

The description of the variants is given in Table 2.6.4 on p. 482.

Additional controls will become available on the Property bar after selection of the required option. The set of these controls and further actions depend on the selected variant.

4. Make additional displacement settings in accordance with the selected variant.
 - ▼ **Along the X axis/Along the Y axis/Along the Z axis** — no additional settings are required.
 - ▼ **Parallel to object/Perpendicular to object** — specify the guiding object in the model window. The name of the object is displayed in the **Guiding Object** field.
 - ▼ **By normal** — enable/disable the **From source** option.

If the option is enabled, the normal's position will be calculated with respect to the edited edge's source form; if the option is disabled — with respect to the current edge's form. The edge's source form is the form that the edge had at the moment of calling the **Spline form** command.



5. Select the type of smoothening by enabling one of the following switches of the **View** group:



- ▼ **Smooth transition,**
- ▼ **Convex,**
- ▼ **Concave.**



6. Enter the function's n degree value in the **Degree** field for the **Convex** or **Concave** variant. The field appears on the Property bar after selection of a variant.



7. Move the selected poles. To do this, move the cursor to one of them. When the symbol of a vertex appears next to the cursor, move the mouse to the required position keeping the left button clicked. After that, release the button. All selected poles (except for the pole which is outermost from the pole which is moved by the mouse) will be displaced in accordance with the preset smoothening type.



8. To finish the command, click the **Create object** button.

2.6.3.1.4. Rotation of poles

Positions of grid poles of the edited face can also be changed by rotation (Fig. 2.6.47).

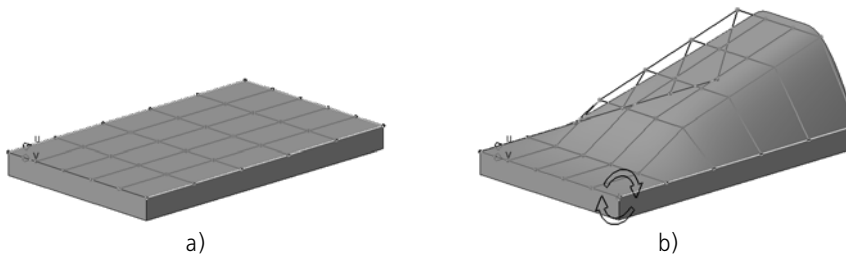


Fig. 2.6.47. Rotation of poles:
a) face's original state, b) rotation result

Grid poles are rotated around the axis going through the rotation center in the defined direction. The rotation plane is perpendicular to the rotation axis and goes through the rotated pole.

An example of the pole displacement by rotation can be found in Fig. 2.6.48. The direction of the rotation axis coincides with the X axis of the current coordinate system; P_{start} and P_{end} — the initial and the final positions of the displaced pole.

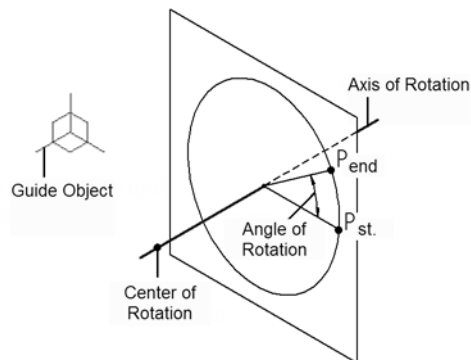


Fig. 2.6.48. Rotation of a pole

In order to move poles by rotation, perform the following operations.



1. Call the **Rotation** command. Round arrows will appear on the screen. The Property bar will display control elements for rotation parameters' settings (Fig. 2.6.49).



Fig. 2.6.49. Property bar in the process of poles rotation



2. Specify the poles that should be rotated (see Section 2.6.3.1.1 on p. 483).
3. Set the rotation axis direction by selecting one of the following variants from the **Direction** drop-down list:
 - ▼ **Along the X axis,**

- ▼ **Along the Y axis,**
- ▼ **Along the Z axis,**
- ▼ **Parallel to an Object,**
- ▼ **Perpendicular to an Object.**

The description of the variants is given in Table 2.6.4 on p. 482.

4. Specify the rotation center. To do so, select one of the following variants in the **Center** drop-down list:

- ▼ **None,**
- ▼ **LCS,**
- ▼ **Object's center,**
- ▼ **Center of selected,**
- ▼ **Point.**

The description of the variants is given in Table 2.6.5.

Tab. 2.6.5. Variants of the rotation center setting

Variant	Description
No	No center is specified. The rotation axis coincides with the guiding object or normal to the object (for direction Perpendicular to object).
LCS	The center coincides with the beginning of the current coordinate system (see Section 2.8.3.3 on p. 597).
Object's center	The center coincides with the mass center of the edge which is being edited. Coordinates of the mass center is calculated for the form the edge has at the moment of editing.
Center of selected	The center coincides with the mass center of the selected poles. To calculate mass center, the poles are presented by material points of similar weight.
Point	If this variant is selected, the Center's object switch is located on the Property bar (it is enabled by default). This switch makes it possible to specify a custom point object including one of the grid poles. The name of the selected object is displayed in a field next to the switch. The center will coincide with the object.

5. Enter the angle of revolution value into the **Angle** field. It can be both positive — rotation in the direction of the arrows — and negative with rotation in the opposite direction.

The selected grid poles will be moved in accordance with the preset rotation parameters.

6. If necessary, you can specify the manually selected objects again — grid poles or objects which define rotation parameters. For this purpose, enable the required switch (for example, the **Selection of poles** switch should be enabled for specification of a pole) and specify the object. To redefine all manually selected objects, click **Reselect** button of the Special Control Panel and make your choice again by enabling the respective switches.





7. To finish the command, click the **Create object** button.

2.6.3.1.5. Scaling

Positions of grid poles of the edited face can also be changed by scaling.

In order to displace a pole by scaling, the following methods are used (Fig. 2.6.50):

- ▼ even scaling,
- ▼ directional scaling,
- ▼ scaling in a plane.

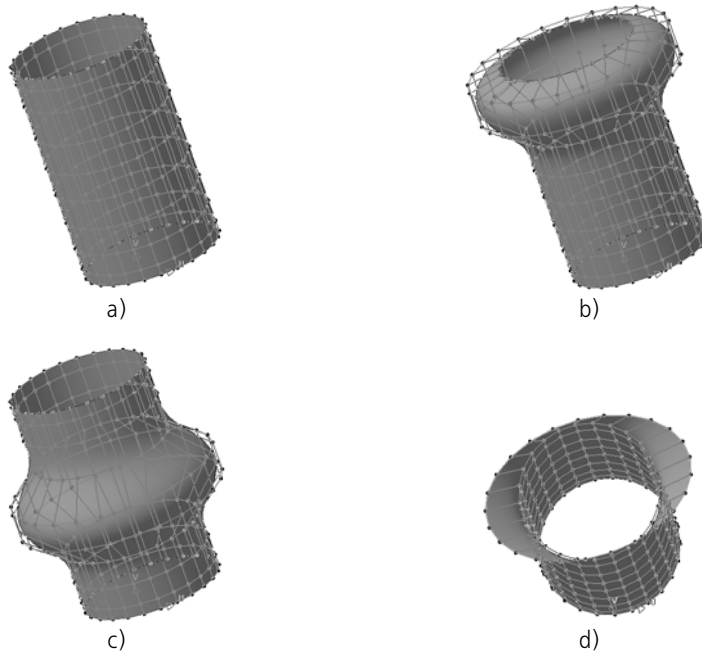


Fig. 2.6.50. Pole scaling:
a) face's original state, b) even scaling,
c) scaling in a plane, d) axis scaling

The scaling is done as follows.

Even scaling — the pole is moved along a straight line connecting the scaling center with this pole until the distance between it and the scaling center increases k times, where k is the scaling factor.

An example of the pole displacement by even scaling can be found in Fig. 2.6.51.

This is a custom scaling center.

$P_{init.}$ and $P_{end.}$ — the initial and the final positions of the displaced pole.

$L_{init.}$ and $L_{end.}$ — distances between the initial and the final positions of the pole and the scaling center.

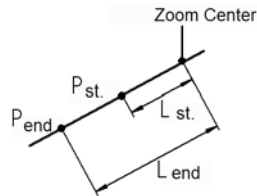


Fig. 2.6.51. Pole displacement with even scaling

Directional scaling — the pole is moved parallel to the straight line going through the scaling center in the defined direction until the distance between its projection on that straight line and the scaling center increases k times, where k is the scaling factor.

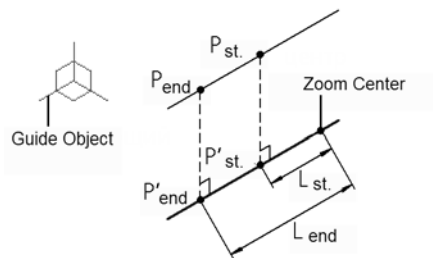
An example of the pole displacement by directional scaling can be found in Fig. 2.6.52.

The displacement direction coincides with the X axis of the current coordinate system, i.e. the pole is displaced parallel to the straight line going through the scaling center parallel to the X axis. This is a custom scaling center.

P_{init} . and P_{end} . — the initial and the final positions of the displaced pole.

P'_{init} . and P'_{end} . — projections of the initial and the final positions of the displaced pole on a straight line parallel to which the pole is being displaced.

L_{init} . and L_{end} . — distances between the initial and the final positions of the pole projections and the scaling center.

Fig. 2.6.52. Pole displacement during directional scaling, the X axis is the guiding object.

Scaling in a plane — the pole is moved parallel to the straight line connecting the scaling center and the pole's projection on the plane going through the scaling center parallel to the preset flat object. The displacement is performed until the distance between the pole's projection and the scaling center increases k times, where k is the scaling factor.

An example of the pole displacement by scaling in a plane can be found in Fig. 2.6.53.

The scaling plane is parallel to the XY plane of the current coordinate system and goes through the scaling center. This is a custom scaling center.

P_{init} . and P_{end} . — the initial and the final positions of the displaced pole.

$P'_{init.}$ and $P'_{end.}$ — projections of the initial and the final positions of the displaced pole on the scaling plane. The straight line, parallel to which the pole is moved, goes through the scaling center and the projection of the initial position of the $P'_{init.}$ pole.

$L_{init.}$ and $L_{end.}$ — distances between the initial and the final positions of the pole projections and the scaling center.

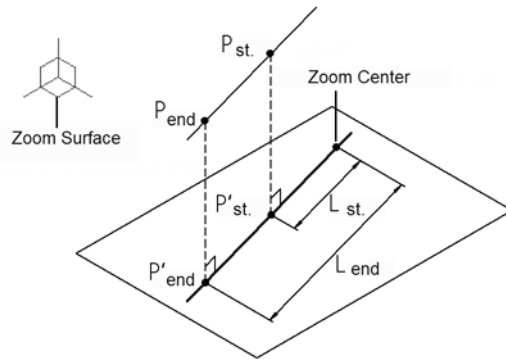


Fig. 2.6.53. Pole displacement during scaling within a plane, XY is the scaling plane.

In order to move poles by scaling, perform the following operations.



1. Call the **Scaling** command. The Property bar will display control elements for scaling parameters' settings (Fig. 2.6.54).

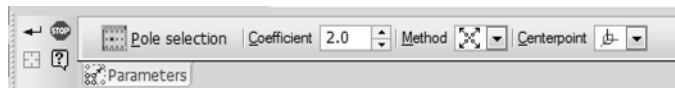


Fig. 2.6.54. Property bar during the displacement of poles by scaling



2. Specify the poles that should be displaced (see Section 2.6.3.1.1 on p. 483).
3. In the **Coefficient** field, enter the scaling coefficient. Selected poles will be displaced in accordance with the preset coefficient in the direction which is defined by the default scaling method and center.
4. Select the required scaling method from the **Method** drop-down list. The following s are available.
 - ▼ The **Even** variant for even scaling (selected by default).
 - ▼ For directional scaling:
 - ▼ **Along the X axis,**
 - ▼ **Along the Y axis,**
 - ▼ **Along the Z axis,**
 - ▼ **Parallel to an Object,**
 - ▼ **Perpendicular to an Object.**

The description of the variants is given in Table 2.6.4 on p. 482.

- ▼ For scaling within a plane:
 - ▼ **Within the XY plane,**
 - ▼ **Within the ZX plane,**
 - ▼ **Within the ZY plane,**
 - ▼ **In selected plane.**

If one of the first three variants is selected, its respective coordinate plane of the current coordinate system will be selected automatically (see Section 2.8.3.3 on p. 597).

The **In selected plane** variant makes it possible to specify a custom flat object (see Table 2.1.7 on p. 131) in the Design Tree or Model Window. There is also the Guiding object **switch (enabled by default) on the Property bar**. Specify the required object. The name of the object will be displayed in the field beside the switch.



5. Specify the scaling center. To do so, select one of the following variants in the **Center** drop-down list:
 - ▼ **LCS,**
 - ▼ **Object's center,**
 - ▼ **Center of selected,**
 - ▼ **Point.**

The description of the variants is given in Table 2.6.5 on p. 493.

After setting all scaling parameters, the selected poles will be positioned appropriately.

6. If necessary, you can specify the manually selected objects again — grid poles or objects which define scaling parameters. In order to specify the required object, activate the related switch (for example, the **Pole selection** switch should be enabled for pole specification. To redefine all manually selected objects, click **Reselect** button of the Special Control Panel and make your choice again by enabling the respective switches.
7. To finish scaling, click the **Create object** button.



2.6.3.1.6. Pole alignment

Positions of grid poles of the edited face can also be changed by alignment.

Two methods of grid poles alignment are available: **By direction** (Fig. 2.6.55) and **By flat object** (Fig. 2.6.56).

In the first case, the selected pole is moved to a straight line going through the alignment center in a predefined direction. The poles are moved perpendicular to the straight line.

In the second case, the selected pole is moved to the plane parallel to the predefined plane and going through the alignment center. The poles are moved perpendicular to the plane.

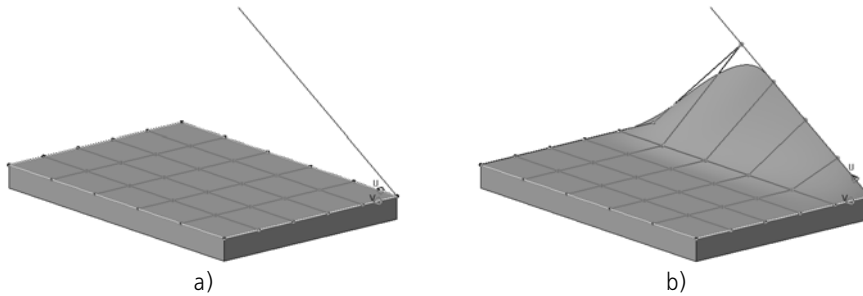


Fig. 2.6.55. Alignment of poles by direction:
a) face's original state, b) rotation result

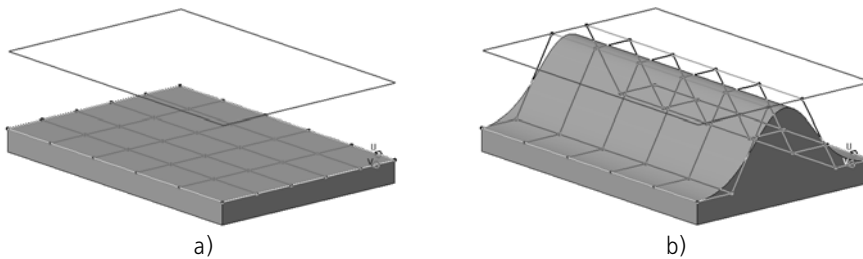


Fig. 2.6.56. Alignment of poles by a flat object:
a) face's original state, b) rotation result

To move the pole by alignment (row of poles), complete the following steps.



1. Call the **Alignment** command. The Property bar will display control elements for alignment parameters' settings (Fig. 2.6.57).



Fig. 2.6.57. Property bar during the displacement of poles by alignment.



2. Specify the poles that should be displaced (see Section 2.6.3.1.1 on p. 483). The selected poles will be moved in accordance with the preset alignment parameters.
 3. Select the required alignment method from the **Method** drop-down list.
 4. Set direction or alignment plane.
- ▼ For alignment by direction, set the direction using the drop-down list with the same name. The following methods are available:
 - ▼ **Along the X axis,**
 - ▼ **Along the Y axis,**
 - ▼ **Along the Z axis,**
 - ▼ **Parallel to an Object,**

▼ **Perpendicular to an Object.**

The description of the variants is given in Table 2.6.4 on p. 482.

- ▼ For alignment by direction, set the direction using the drop-down list with the same name. The following methods are available:
 - ▼ **XY plane,**
 - ▼ **ZX plane,**
 - ▼ **ZY plane,**
 - ▼ **Selected plane.**

If one of the first three variants is selected, its respective coordinate plane of the current coordinate system will be selected automatically (see Section 2.8.3.3 on p. 597).



The **Selected plane** variant makes it possible to specify a custom flat object (see Table 2.1.7 on p. 131) in the Design Tree or Model Window. There is also the Guiding object **switch (enabled by default) on the Property bar**. Specify the required object. The name of the object will be displayed in the field beside the switch.

5. Specify the alignment center. To do so, select one of the following variants in the **Center** drop-down list:
 - ▼ **None,**
 - ▼ **LCS,**
 - ▼ **Object's center,**
 - ▼ **Center of selected,**
 - ▼ **Point.**

If the **No** variant is selected, no alignment is set. In the alignment process, straight line (plane) to which the poles are moved coincides with the selected straight-line (flat) object or normal to the object (for direction **Perpendicular to object**).

Description of other options for center setting is presented in Table 2.6.5 on p. 493.

After setting all alignment parameters, the selected poles will be positioned appropriately.

6. If necessary, you can specify the manually selected objects again — gird poles or objects which define alignment parameters. In order to specify the required object, activate the related switch (for example, the **Pole selection** switch should be enabled for pole specification. To redefine all manually selected objects, click **Reselect** button of the Special Control Panel and make your choice again by enabling the respective switches.



7. To finish alignment, click the **Create object** button.

2.6.3.2. Pole fixation

In the process of face editing, you may need some poles to remain static — fixed.



In order to fix poles, call the **Fix poles** command on the **Spline form** toolbar. The Property bar will display control elements for fixation settings (Fig. 2.6.58).



Fig. 2.6.58. Property bar in the process of poles fixation

The following fixation methods are available:

- ▼ Fixation of poles located on the grid borders (in its outermost rows),
- ▼ Custom pole fixation,
- ▼ Fixation of poles ensuring constant form and position of a face's edges.

You can fix poles using one or several methods at the same time. The procedure of pole fixation is described below.



To finish poles' fixation, click the **Create object** button.

Fixed poles are displayed on the screen as circles with a larger diameter.

2.6.3.2.1. Fixation of poles on the grid borders

The poles located on the grid borders are fixed using the group of switches **Borders along U** (for grid borders located in the U direction) and **Borders along V** (for grid borders located in the V direction) as is shown in Fig. 2.6.58.

By default, none of the grid borders is fixed, therefore, **No fixation** switches are enabled in the groups. Use the following switches to fix the required borders' poles:

- ▼ **Both grid borders**,
- ▼ **First grid border**,
- ▼ **Second grid border**.



In the face edit mode, you can fix all grid borders or cancel their fixation without calling the **Fix poles** command. To do so, the **Fix grid borders/Cancel fixation of grid borders** context menu command is used. Once this command has been called, switches for border fixation are automatically enabled in the **Fix poles** command:

- ▼ the **Fix grid borders** command — **Both grid borders** switches,
- ▼ the **Cancel fixation of grid borders** command — **No fixation** switches.

2.6.3.2.2. Custom fixation of poles

If you need to fix grid poles with free positions, enable the **Selection of poles** option and specify the required poles or pole rows by the mouse (for more information about pole selection, please refer to Section 2.6.3.1.1 on p. 483). Poles may be located in any grid row and on its border as well.

You can cancel fixation of a certain pole (pole row) by another click of the mouse on that pole (pole row). To cancel fixation of all poles, which had been fixed using this method, disable the **Selection of poles** option.



In the face edit mode, you can enable or disable selection of poles for fixation without calling the **Fix poles** command. For this purpose, the **Fix poles with Shift** context menu command is used. Calling this command makes it possible to select poles for fixation with the <Shift> key pressed; repeated calling cancels the fixation. In this case, the status of the **Pole selection** option in the **Fix poles** command will be changed automatically.

2.6.3.2.3. Fixation of poles ensuring constant form and positions of a face's edges

In the process of face editing, you may need to have the form and position of some or all of its edges unchanged. For this purpose, you can fix grid poles affecting the form and position of those edges.

For this kind of fixation, enable the **Selection of face's edges** option and specify all required edges by clicking on them one by one.

The selected edges are highlighted; all grid poles affecting the form and the position of these edges will be fixed automatically.

In order to cancel fixation of poles ensuring constant form and position of one of the selected edges, click this edge again. If you need to cancel fixation of poles for all selected edges, disable the **Selection of face's edges** option.

2.6.3.3. Changing spline surface parameters

In order to change the form of a face transformed into a spline surface (see Section 2.5.1.4 on p. 329), the grid formed by rows of control points (poles) is used. The grid poles are located in two directions: U and V.

By default, the grid has 6x6 dimensions (i.e. the number of grid poles along each direction is 6) and surface order equal to 4.



If a customized face is initially a spline surface, the grid dimensions and surface order are equal to the face's dimensions and order, respectively.



You can change the surface order and the number of its poles using the following **Spline form** toolbar commands:



▼ **Order**,



▼ **Grid**.

Detailed description of the commands usage can be found below.

2.6.3.3.1. Surface order



In order to change surface order, call the **Order** command.

Order along U and **Order along V** fields will appear in the Property bar. These fields make it possible to set the required order value for each of the grid directions (Fig. 2.6.59).

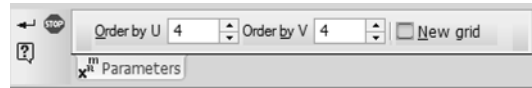


Fig. 2.6.59. Property bar after the grid order change

Enter values in these fields and or set them using the counter. The field's value can not be less than 3.

You can also change the number of grid poles. To do this, enable the **New grid** option. Additional controls will become available on the Property bar (Fig. 2.6.60).

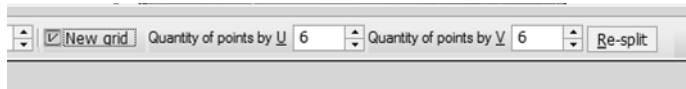


Fig. 2.6.60. Setting the number of grid poles

Enter the required values in **Number of points in U** and **Number of points in V** fields and click the **Split again** button. The displayed grid will be rebuilt. The number of its poles will correspond to the set values.



In the same way, you can change the number of grid poles when working with the **Grid** command.



Note that the number of grid poles along the selected direction cannot be less than the surface's order set for this direction. Therefore, order increase results in automatic increase in the number of poles. Order reduction does not change the number of poles.



To finish the command, click the **Create object** button.



The **Interrupt command** makes it possible to cancel the changes made.



3.3.2. Grid dimensions

In order to make grid denser or wider, i.e. to change its dimensions, call the **Grid** command. Grid setup elements will appear on the Property bar (Fig.2.6.61).

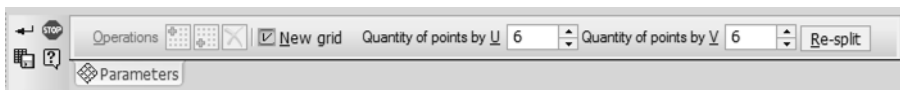


Fig. 2.6.61. Property bar after the grid dimensions change

You can change grid dimensions in two ways: by adding or deleting its rows; by setting the required number of grid poles.



In order to add a new row, specify one of the grid rows. In order to do so, click the grid's segment between two poles of the required row. In the **Operations** group, click the **Insert before the**



selected row or **Insert after the selected row** button. The new row will be positioned in the same direction as the selected row.



In order to delete a row, click it and then click the **Delete the selected row** of the **Operations** group.

You can also change the number of grid poles using the **New grid** option. Additional controls will become available on the Property bar after enabling the option.

Enter the required values in **Number of points in U** and **Number of points in V** fields and click the **Split again** button. The displayed grid will be rebuilt. The number of its poles will correspond to the set values.



In the same way, you can change the number of grid poles when working with the **Order** command.



If necessary, you can save grid poles' coordinates to the file of data. To do so, click the **Save to file** button on the Special Control Panel and specify the file name and the folder in the appearing dialog. By default the file shall be assigned the *txt* extension. For more information about the data file, see Chapter VI.



To complete grid setup, click the **Create object** button.



The **Interrupt command** button makes it possible to cancel the changes made.

2.6.3.4. Custom displacement of poles by the mouse

In the face editing mode, you can move one or several grid poles by the mouse and also fix poles that need not to be moved.

Pole fixation and displacement methods are described below.

If necessary, you can cancel earlier operations or redo them (see Section 1.4.1.6 on p. 79).

2.6.3.4.1. Techniques

In order to displace poles by the mouse, the following methods are used.

- +*
▼ To displace a pole, move the cursor to it. When the symbol of a vertex appears next to the cursor, move the mouse to the required position keeping the left button clicked. After that, release the button.
- +₂
▼ To displace all poles of one of the grid rows, move the cursor to this row's segment. When the symbol of a 3D curve appears next to the cursor, move the mouse to the required position keeping the left button clicked. After that, release the button.
- ▼
To displace several poles, click the mouse on them holding down the <Ctrl> key. Similar highlighting of a grid segment adds to the set of selected poles all poles in a row containing the segment. Another mouse click on the selected pole (pole row) with pressed <Ctrl> key deselects the pole (pole row).
- ▼
In order to displace all grid poles, highlight them using the **Select all poles** context menu command or <Ctrl>+<A> key combination. The **Cancel pole selection** context menu command deselects all grid poles. If you need to deselect only some of them, click the required poles (pole rows) holding down the <Ctrl> key.

if necessary, you can invert grid pole selection by calling the **Invert pole selection** context menu command. The highlighting will be removed from selected poles; all other poles will be highlighted.

When displacing poles, the form and/or the position of the face will change. If the new position of poles does not ensure correct changing of the face's form or position, the face remains in its initial state (refer to Section 2.6.3.6 on p. 507 for more details).

Note that you can displace unfixed grid poles only. Fixed poles cannot be displaced, even if they are highlighted.

To do so, the following context menu commands are used.

- ▼ **Fix borders** — makes it possible to fix all poles positioned on the grid's borders. To remove fixation, the **Cancel border fixation** command is used.
- ▼ **Pole fixation mode** — you can fix custom grid poles. Once the command has been called, poles are selected by clicking the mouse holding down the <Shift> key. Similar highlighting of a grid segment adds all poles in a row containing the segment to the set of fixed poles. If you need to cancel fixation of a pole (pole row), click it again keeping the <Shift> key pressed. In order to cancel fixation of all grid poles, call the **Pole fixation mode** command again.



You can fix poles or remove fixation using the **Fix poles** command of the toolbar (see Section 2.6.3.2 on p. 499).

By default, the selected poles are displaced within the screen plane. You can set another displacement direction using the **Offset direction** group of commands in the context menu. All available displacement variants can be found in Table 2.6.6.

Tab. 2.6.6. Context menu commands

Command	Description
Along the X axis Along the Y axis Along the Z axis	The commands make it possible to move selected poles along X, Y and Z axis of the current coordinate system (see Section 2.8.3.3 on p. 597).
Parallel to Object	You can use this command to displace selected poles parallel to a straight-line object (see Table 2.1.7 on p. 131) or cylinder, cone, toroid axis. Once the command has been called, specify the required object.
Perpendicular to Object	You can use this command to displace selected poles perpendicular to a flat object (see Table 2.1.7 on p. 131). Once the command has been called, specify the required object.
By Normal	You can use this command to displace selected poles along the normal to the edited face. It is impossible to move the poles behind the row's segment, if this variant is selected.

Tab. 2.6.6. Context menu commands

Command	Description
Along polygon	You can use this command to displace selected poles along U or V directions of the grid. Two offset variants are available for each direction. To the final , To the previous . In the first case, the pole is moved along the grid segment connecting this pole with the next pole in the selected direction (U or V); in the second case — in the opposite direction.
Within the screen plane	This command makes it possible to move selected poles within the screen's plane.

2.6.3.4.2. Peculiarities of setting the pole displacement direction

In case of a custom displacement of poles by the mouse, the displacement direction is set by context menu commands. These commands are related to Property bar controls that are used to set direction when working with **Linear displacement** and **Displacement with smoothening** commands.

For example, the following context menu command was called in the custom pole displacement mode: **Offset direction — Along polygon — Along U — To the previous**. Then, it was required to move poles by the **Linear displacement** toolbar command. After calling this command, the specified direction remained unchanged and its respective controls appear in the Property bar:



- ▼ in the **Direction** list — the **Along polygon** variant,
- ▼ in the **Type** group — **Along U** switch,
- ▼ in the **Method** group — **To the previous** switch.



If the offset direction changes while working with the **Linear displacement** or **Displacement with smoothening** command, this change will be saved after completing the command.



For example, when working with the **Displacement with smoothening** command, the **Parallel to object** variant was selected in the **Direction** list, and the guiding object is specified in the model window. Once working with the command has been finished, the specified direction remains unchanged, while the following command becomes automatically enabled in the context menu: **Offset direction — Parallel to object** (a tick appears next to the command's name). Besides that, the selected guiding object remains unchanged.

2.6.3.5. Construction of a spline form on a face's equidistant

Spline form can be constructed on a surface which is equidistant to the customized face, i.e. the resulting spline form will be a separate surface.

Using an equidistant surface makes it possible to prevent changing the initial face and its adjacent faces.



In order to create a spline form on equidistant surface, click the **Face's equidistant** button on the Special Control Panel after selecting a face for editing. A spline surface equidistant to the edited face will be created (for more information about the spline surface, refer to Section

2.5.1.4 on p. 329). It coincides with the edge, i.e. the distance between the edge and its equidistant is 0.

Note that the face selected for editing will be automatically transformed into a spline surface. After creation of the equidistant, the face will return to its initial state.



The **Face's equidistant** button can also be clicked before face selection. In this case, the equidistant surface will be created after face selection.

The equidistant surface can be edited using the same methods that are used for face editing. If the **Face's equidistant** button is enabled, all actions in the face editing mode are applied to its equidistant while the form and the position of the source face remain unchanged (Fig. 2.6.62).

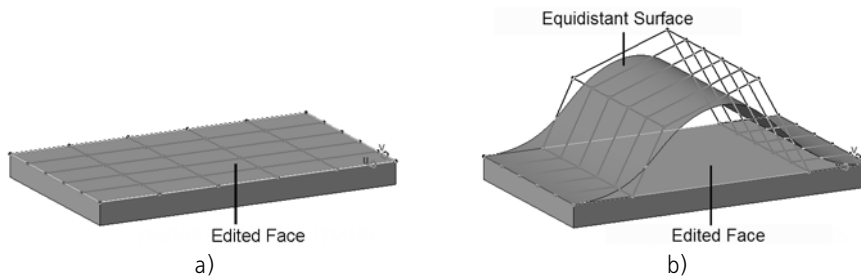


Fig. 2.6.62. Construction of a spline form on equidistant surface:
a) customized face,
b) equidistant surface and customized face



If a face had been customized prior to creation of an equidistant surface, the equidistant surface will take the form and the position of the face, which is being customized at the moment, while the customized face will return to its initial form and position.

You can cancel spline form creation on the equidistant of the edited face. To do so, disable the **Edge's equidistant** button. The equidistant surface will disappear while the customized face will be transformed into spline surface again. All operations performed in the editing mode will be applied to the edited face. The face will take the equidistant's form and position available at the moment of editing.



After editing the equidistant, its borders may differ from the source face's borders and be positioned in such a way that it will not be possible to join them with the adjacent faces of the part. In this case, when going back to editing the face, its form and position may differ from the equidistant's form and position at the moment. The face will remain in its initial state, as well as the grid's ghost.

Detailed description of face and face's equidistant customization peculiarities can be found in Section 2.6.3.6 on p. 507.

2.6.3.6. Face and face's equidistant customization peculiarities

- ▼ Note the following specifics when editing a face.
 - ▼ If the customized face is flat, its form will not change when moving grid poles within the face's plane.
 - ▼ In the process of face editing, the form and the position of its border may change. The form and the position of its adjacent face border may also change. These changes are possible only if the type of the adjacent face remain unchanged.

For example, the edited face and its adjacent face are both flat. In order to change the form of the edge between those faces, you should move the grid poles on it. You can move the poles in different directions. However, in this case, the edge's form will be changed in the plane of the face which is adjacent to the customized face. If poles' positions make it impossible to create an edge in this plane, the form of the customized edge will return to its initial state. And the grid poles will remain in the same positions.
- ▼ Customization of a face or its equidistant has the following peculiarities.
 - ▼ If the initial face contains more than 1000 edges, the **Spline form** command will be performed a bit slower.
 - ▼ If grid poles' positions were changed in the process of editing in such a way that the form of the face/face's equidistant cannot be built correctly, the face/face's equidistant will return to its initial form. And the grid poles will remain in the same positions.
 - ▼ Self-intersection is acceptable in the process of face/face's equidistant editing.
 - ▼ If you begin to edit a face's equidistant after editing the face, the face's current form and position will be applied to the equidistant. It is the same for the reverse situation (editing a face after editing its equidistant), except for cases when equidistant borders have the same position when joining the adjacent faces of the part is impossible. In this situation, the face will have the initial form and position.



The peculiarities of edge/edge's equidistant editing relate not only to the initial spline form creation, but also to its further editing.

2.7. Arrays

2.7.1. General information

2.7.1.1. Info on arrays

When working with a model, it may be necessary to create copies of objects (for example, operations or assembly components) which would be arranged in a certain way, forming a rectangular grid with set-up parameters or being symmetric about a plane.

To create ordered groups made of similar objects in the model, you can use array construction commands. KOMPAS-3D supports the capability of creating the arrays of the following types:

- ▼ by grid (see Chapter 2.7.3 on p. 537),
- ▼ by circular grid (see Chapter 2.7.4 on p. 543),
- ▼ along a curve (see Chapter 2.7.5 on p. 553),
- ▼ by Points (see Section 2.7.6 on p. 560),
- ▼ by table (see Chapter 2.7.7 on p. 565),
- ▼ mirror array (see Chapter 2.7.8 on p. 575),
- ▼ by sample (a kind of assembly components array; see Chapter 2.7.9 on p. 578).

Commands for constructing arrays are located in **Operations** menu, and the buttons to call them are on the **Arrays** panel (Fig.2.7.1).



Fig. 2.7.1. Geometric Objects Copy Commands

You can create arrays of the following objects:

- ▼ operations adding or deleting a body's material;
- ▼ bodies,
- ▼ surfaces,
- ▼ of curves,
- ▼ pixels,
- ▼ auxiliary planes,
- ▼ auxiliary axes,
- ▼ components.

You can copy previously created point groups, groups of isoparametric curves and arrays (except for arrays of bodies and surfaces and arrays with the table of modifiable variables).

The array of objects consists of **instances**. By default the first array instance is a **base instance**. It corresponds to the object being copied or to a group of objects, if there are several objects to be copied. Other array instances are copies of the base instance. You can change the array instances' position with respect to the base instance by assigning a certain number to it in the array.

If necessary, you can change the form and dimensions of instances by changing their variables' values (provided that the object being copied has variables). Such an option is available for all arrays except for mirror array and array by sample. Managing instance variables is described in detail in Section 2.7.2 on p. 519.

If bodies consisting of parts result from creation of the operations' array, the array's icon has a sign of several parts present— exclamation mark. For more information about bodies consisting of several parts, see Section 2.3.11 on p. 224.

Instances can be excluded from all arrays except for the array by sample (see Section 2.7.1.3.6 on p. 516).

All arrays, except for the operations' array, can be exploded to individual instances (see Section 2.7.1.3.7 on p. 517).

Creating an array by sample has some peculiarities. See Section 2.7.9 on p. 578 for details. Other arrays have common operation methods. See description of these methods in Section 2.7.1.3 on p. 512. See chapters 2.7.3–2.7.8 for more information on creation of these arrays.

2.7.1.1.1. Base point of an array's instance

The **base point of an instance** concept is used in the following arrays:

- ▼ on circular grid,
- ▼ by points,
- ▼ by table,
- ▼ along a curve.

The instance's location is defined with respect to the base point of the instance. At the same time, positions of the instances' base points are defined by type and parameters of the array. In the array on circular grid, the instances' base points will coincide with the grid nodes, in the arrays by points or by table – with the indicated points or with points obtained from a file, while in array along a curve, the instances' base points are positioned on the curve.

The base point of the base instance of an array is the **base point of the object to be copied** (see Section 2.1.1.4 on p. 104). Moreover, it is possible to select a point as a base instance's base point. Eventually, since all instances of the array are copies of the base array, their position with respect to base points will change in a similar way.

More information on building the above-mentioned arrays can be found in chapters 2.7.4 and 2.7.7.



If a rotation operation, a body created by the rotation operation, or a rotation surface is being copied, you can use projection of the **reference point of the object being copied** on the rotation axis of the object being copied instead of the point (see Section 2.7.1.3.3 on p. 514).

2.7.1.1.2. Displaying array's instances in the model's Design Tree

Array instances are displayed in the model's Design Tree as separate objects dependent on the array.

To expand the list of instances, click the mouse on the “plus” sign located at the left of the array icon in the Design Tree.

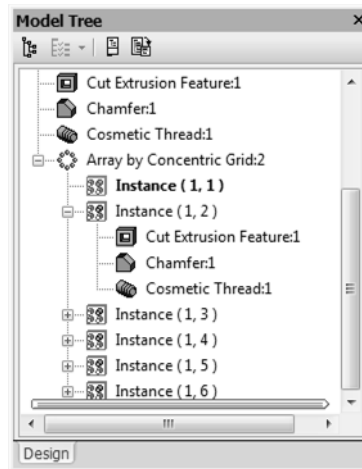


Fig. 2.7.2. Array on circular grid in the Design Tree

An array instance ID number is displayed in parentheses at the right of the instance icon.

- ▼ In case of a 2D linear array, an array instance ID consists of two numbers. The first one is an instance ID number along the first array axis, and the second one is the instance ID along the second axis (instance numbering starts with 1).
- ▼ For a circular array, an instance ID consists of two numbers. The first one is an instance ID number along the radial direction, the second one is the instance ID along the circumferential direction (instances numbering starts with 1).
- ▼ For copies placed along the curve, an instance's ID number is counted in the order the instances follow, starting from the source one.
- ▼ If copies are placed by points, the instance's number is the same as the number of the point which determines its position. The points are numbered in order of their selection.
- ▼ If copies are placed by coordinates (specified in a file or table), the instance's number is equal to serial number of the line in the file or table.

2.7.1.1.3. Point Variables

When you create an array in the Variables window of the model, the system automatically creates a section to store the variables of this array.

For the variables of each instance of the array, the system creates a dedicated subsection nested into the section of array variables. For example, the *Array by table:1* variables section can contain subsections *Instance (1)*, *Instance (2)* and so on.

Array variables include variables of sketch dimensions and operations. Such an option is available for all arrays except for mirror array and array by sample.

For an array by table, the instance variables also include the variables of instance dimension positions. These variables are automatically created when you set the positions of array instances. They correspond to the point coordinates read from the data file or can be specified in the table of array variables (see Section 2.7.7.2 on p. 568).

2.7.1.2. Application area of the operation's array instance

Each array's instance inherits the application area of the operation which it copies. For example, each copy of a rib is attached to the same body, to which the initial rib had been attached or each copy of a circular hole is cut from the same body or component, from which the initial hole had been cut and so on.

Changing of the ranges of applicability of operations' array instances is possible neither while creating, nor while editing the array.



When required, you can create an array of material removal operations (e.g. holes) to apply each copy of this array to the copy of the component. To do this, you can do the following:

- ▼ Create a component group.
- ▼ Build a hole going through the original component and add then add the component copies though which the copies of the hole must go through into the usage area.
- ▼ Create an array of holes.

2.7.1.3. General method of working with arrays

2.7.1.3.1. Choice of objects for copying

To choose objects, use the **Object selection** tab on the property tab (Fig. 2.7.3).



Fig. 2.7.3. **Object Selection** tab

The switches of the **Type** group enable the selection of the copied objects:



- ▼ **Auto definition by a setting;**



- ▼ **Operations;**



- ▼ **Bodies or surfaces;**



- ▼ **Curves, points, auxiliary geometry,**



- ▼ **Arbitrary** — this switch is available in the **Type** group only when an array by table is being created; for details on how to build an array with this switch enabled see Section 2.7.7.1 on p. 565.



- ▼ **Components**

By default, the **Auto definition by a setting** switch is enabled in the group. This means that the object type will be defined by the system automatically after setting the first object; the switch matching the type of the set object is enabled in the group. Specified objects becomes color highlighted, and its name will be displayed on the **List of objects** bar. To cancel object selection, re-select it.

If needed, select other objects of this type as objects for copying.

Note that the curves, points, auxiliary planes and axes can be bulk copied (i.e. can be located in the same array) while the bodies and surfaces cannot (i.e. an array can contain either bodies or surfaces).



During the creation of mirror copy of body or surface, the **List of Objects** panel is not active on the Properties bar.

To select objects of a certain type for copying, enable the switch of the respective type in the **Type** group. Then specify selected type object(s) in the Design Tree or in the model window.



Please note that when the edge, face or body vertex is selected in the model window, the **Operations** switch activates in the **Type** group, and when the edge, face or body vertex from the component contents is selected—the **Component** switch. If you need to select a body to copy, then activate **Bodies or Surfaces** switch before its selection.



You can also select objects for copying prior to calling the array creation command.



When copying the components, note that only the components of the first level (i.e. inserted directly into the current model) can be copied. Therefore, if an object is set in the Design Tree or model window, which is included in a component, the component will be selected for copying and its name will appear on the **List of objects** bar.

If a previously created array is used as an object for copying, it will be copied in its current condition. To copy an array, specify it in the Design Tree and specify any instance of the array in the model window. The name of the specified array will appear on the **List of objects** bar. Note that the type of objects of the array to be copied should be the same as the selected type of objects.



Copying body or surface arrays is impossible. Therefore, these arrays cannot be selected as source objects for an array.

A group of points of isoparametric curves can be copied both as a whole and partially. To copy the whole group of points or curves, specify this group in the Design Tree. The name of the specified group will appear on the **List of objects** bar. To copy individual points or curves, specify the required points or curves in the Design Tree or model window.

Selection of objects in the Design Tree or window is described in detail in Chapter 2.1.4.1 on p. 130.



To cancel selection of all the objects of the set type click **Re-select** button on the Advanced Control panel.

Before completion of the command the system “remembers” the objects specified for copying, so you can switch between function types without loss of their data.



Selection of copied object type is only available when creating an array. It is impossible to change the copied object type during array editing.

Peculiarities of selection objects for copying are described in Section 2.7.1.3.2.

2.7.1.3.2. Peculiarities of selection objects for copying

Chamfer and Fillet operations

Chamfer and **Fillet** operations can only be copied together with the object to which they had been applied. To copy a chamfer or a fillet, specify object creation operation in the Design Tree so that a chamfer or a fillet are built on the object's edges and then specify the chamfer or the fillet. In the model window, you can also specify edges, faces or vertices of the objects to be copied.

Rib

The rib array will be created correctly, if each copy of the rib adjoins the face of the body which the copied rib adjoins as well.

Shell and Slope operations

Shell and **Slope** operations cannot be copied using the array creation commands. Therefore, you should at first create copies of the required object, and then apply **Shell** and **Slope** operations to each copy.



You can copy an element without changing the slope of its faces (**Slope** command). To do this, enable the **Geometrical object** option when creating array of the element (see Section 2.7.1.3.5 on p. 515).

Add Thickness operation

If a new body is created using the **Add Thickness** operation, you can create a copy of this operation.

If the **Add Thickness** operation is applied to an existing body, it is impossible to copy it. Therefore, you should at first create copies of the required body, and then apply **Add Thickness** operation to each copy.

Cosmetic thread

If, upon copying a cosmetic thread, the feature whose face borders this thread is also being copied, the border for copied thread will be the copied face. Otherwise, the copied thread will be bound by the same face as the source one.

2.7.1.3.3. Location variants for the reference point of the object being copied for rotation bodies and surfaces

Instead of the **reference point of the object being copied** (see Section 2.1.1.4 on p. 104), you can use its projection on the rotation axis of the object being copied.

This option is available in arrays that use the **reference point of the base instance** (see Section 2.7.1.1.1 on p. 510) when copying the following objects:

- ▼ rotation operation,

- ▼ body created by the rotation operation,
- ▼ rotation surfaces.

You can enable/disable the use of the reference point's projection in the **Parameters** tab of the Property Bar using the **On rotation axis** option.

By default, reference point's projection is used for array creation, i.e. the **On rotation axis** option is enabled.

A change of the option's status is immediately reflected on the pattern ghost in the Model window.

2.7.1.3.4. Enabling/disabling display of array's ghost



When creating an array, you can enable or disable display of its ghost on the screen, as needed. Such an option is available for all arrays except for mirror array and array by sample.

To manage display of the ghost for the current (created or edited) array, button **Display ghost** is used in the Special Control Panel.

Upon changing the array settings, its ghost dynamically updates.

Using ghost makes array creation more convenient, since that supports a visual inspection of the creation result.

2.7.1.3.5. Geometric array

While creating an operation array, it is possible to expedite array creation and reconstruction. To do this, enable the **Geometric Array** option on the **Parameters** tab. When creating geometric array, only those faces and edges will be copied which result from the operation to be copied. No copying of operations and their parameters is done.

For the geometrical array to be constructed correctly, it is required that all its source objects were constructed either by adding, or by cutting material. You cannot mix objects of different types for copying.



The position of the copied surfaces with respect to existing ones which does not form a body, or the position of array's instances which does not result in an intersection of faces with each other, results in a construction error. In such cases, you should disable the **Geometrical Array** option.

Note the peculiarities when copying an object extruded **To surface**.

- ▼ When copying an object with the **Geometrical array** option disabled, each array instance is extruded to the same surface as the object to be copied. As a result, instances may differ in the height and shape of the “end face”.
- ▼ When copying an object with the **Geometrical array** option enabled, each instance is an exact copy of the source object.



If an array contains a table of variables (for details on a table, see Section 2.7.2.2 on p. 521), then when the **Geometrical array** option is enabled, you will see a message indicating that all variables will be removed from the table. Once you click **OK** all variables will be removed. For an array by table, all the variables except the coordinates of instance basic points are removed.

2.7.1.3.6. Removing and restoring individual instances

Sometimes, you may need to cancel creation of some instances or, vice versa, restore deleted instances. This can be done by doing the following when creating or editing an array.

1. Activate the **Deleted** tab on the Property Bar. Hot points and array instance numbers will be displayed on the screen.



2. In the **Mode** group, enable the **Delete/restore** switch¹.
 3. In the Model window, place the cursor on a handle of the array instance that needs to be deleted or restored. When the “Delete/Restore Instance” message appears next to the cursor, click the left mouse button.
- ▼ If an instance is deleted, its ghost will disappear, and its number will be displayed in the **Deleted** list on the property bar.
 - ▼ If an instance is restored, its ghost will appear again, and its number will be removed from the **Deleted** list.

You can restore deleted instance in a different way.



1. Highlight it in the **Deleted** list.
2. Click the **Restore** button above the list.



Only an entire instance can be excluded from an array. You cannot exclude individual copies of source objects from the contents of an instance.

There is another method of deleting array instances. Unlike the above described method, it becomes available only after the creation of an array. To modify the array in such a way, do the following.

1. Highlight instances in the Design Tree to be deleted and click the *<Delete>* button or call the **Delete** command from context menu.

A dialog will appear prompting you to specify whether all instances will be deleted (i.e. undoing the entire array creation operation) or only the selected instances will be deleted (Fig. 2.7.4).

2. Turn on **Instances** option.

Numbers of deleted array instances are displayed in the dialog (numbers of array instances are described in detail in Section 2.7.1.1.2 on p. 510).

3. Select the instance and click **OK** in the dialog.

The array will be rebuilt and displayed in the model window without the specified instances.

1. If you are going to copy bodies or surfaces, then the **Mode** group will not be available on the Properties panel, so you can simply skip this step. The same is for copying operations, when the **Geometrical array** option is enabled, and for copying custom selected objects using the **Array by table** command.

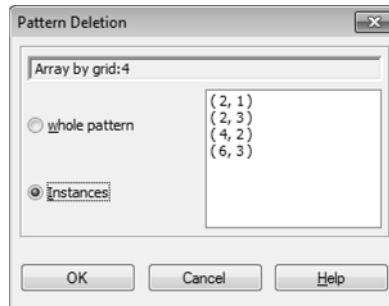


Fig. 2.7.4. Deleting array instances



You cannot delete array instances by sample.

2.7.1.3.7. Explosion of arrays

All arrays, except for operations array, can be exploded to individual instances.

You can do this by selecting the array to be exploded in the Design Tree and using the **Explode** command from the **Editor** menu.



In order to explode a single array, the **Explode** command may be called from the array's context menu in the Design Tree.

Upon calling the command, an explode dialog will appear. To confirm explosion, click **OK** in this dialog.

The rules of exploding arrays are as follows:

- ▼ the exploded array icon and its instance icons are removed from the Design Tree,
- ▼ array's objects are displayed in the Tree as individual model objects,
- ▼ objects which compose the array will change neither their parameters nor their positions in this array;
- ▼ objects corresponding to deleted instances of the exploded array will not be added to the model;
- ▼ if an original array for another (derived) array is exploded, the derived array is also exploded; exploding a derived array does not affect the original array.
- ▼ the components that comprised the array remain at positions they occupied in the array;

The types of objects generated by array explosion depend on the types of the array source objects (Table 2.7.1).



The output of exploding an array of custom selected objects by table (see Section 2.7.7.1 on p. 565) does not match the values in the table 2.7.1. All objects which compose the current array are converted into objects that exactly match the original ones.

Tab. 2.7.1. Results of Exploding an Array of Geometrical Objects

Source Object Type	Types of Objects Resulting from Array Explosion
Body	Body from Nowhere A body from nowhere cannot be edited.
Surface	Surface from nowhere A surface from nowhere cannot be edited.
Points and point groups	Point drawn by the By Coordinates (XYZ) method
3D Lines:	
Line	Segment by two points
Circular Arc	Circle Arc by Three Points.
Polyline	Polyline.
Spline By Vertices	Spline By Vertices.
Spline by Poles, Joining Curves	Spline by Poles.
Other curves	Curve from nowhere Curves from nowhere cannot be edited.
Auxiliary plane	Orthogonal plane (the position of a point through which the plane goes and the direction of the vector to which it is orthogonal are set automatically).
Auxiliary axis	An axis going through a vertex of an object (the position of a point through which the axis goes and the direction of the vector to which it is orthogonal are set automatically).

Note the following specifics of exploding an array of components. Once the array has been exploded, its components are positioned in the Model Tree so as if they were added to the assembly instead of array creation. This can be visualized by turning on model design sequence display in the Model Tree (methods of displaying objects in the Model Tree — see Sections 2.1.2.1 on p. 106 and 2.1.2.2 on p. 106).

For example, the assembly contained the following components added one by one: *Case*, *Gasket*, *Cap* (Fig. 2.7.5, a). Then, the *Gasket* array is created and the *Plug* component is added (Fig. 2.7.5, b). Then the array is exploded. The resulting sequence of components in the Model Tree will be as follows: *Case*, *Gasket* (1), *Cap*, *Gasket* (2), *Gasket* (3), *Plug* (Fig. 2.7.5, c).

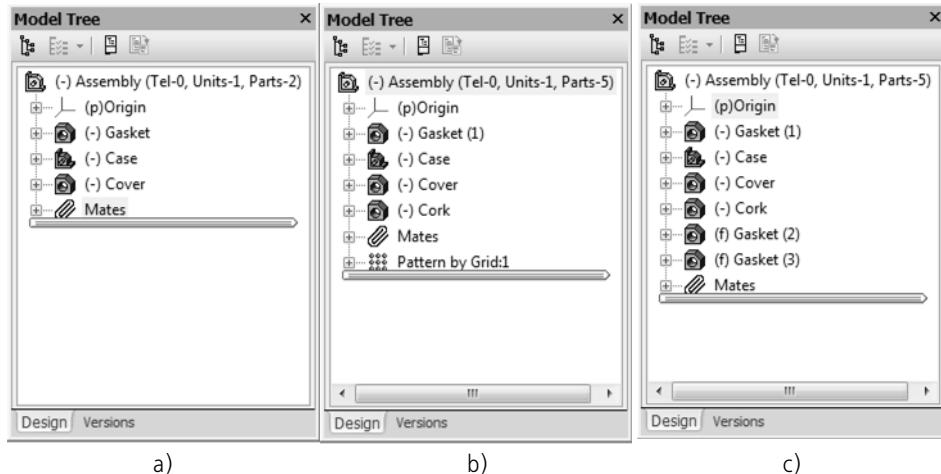


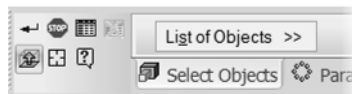
Fig. 2.7.5. Creating and destroying component arrays



If you've changed the values of the array instance variables (see Section 2.7.2 on p. 519), then after the explosion of such an array, the instance's shape and dimensions corresponding to the updated values will not change.

2.7.1.3.8. Peculiarities of array editing

When editing an array, it is impossible to change the type of the object being copied. The **Type** group of switches is not available on the **Choice of Objects** tab. This tab displays the icon of the object type selected for copying during array creation (Fig. 2.7.6 on p. 519).

Fig. 2.7.6. **Choice of objects** tab when editing an array of curves or points

2.7.2. Managing instances of variables

2.7.2.1. General Information

You can change the form and dimensions of instances by changing their variables' values. This option is available provided that the object being copied has variables affecting its form and dimensions.

Instances have the same variables as the object being copied. For example, variables of the extrusion operation are as follows: distance, tilt, and thickness of an item (for thin-walled items). These variables are also available in the instances of extrusion arrays.

By default, variables' values of the object being copied and instances are the same. You can create new values of variables for all array instances, except for the base one.

In Fig. 2.7.7, you can find an example of changing the dimensions of array instances along a curve. The extrusion operation is the copied object. New extrusion distance is set for each instance.

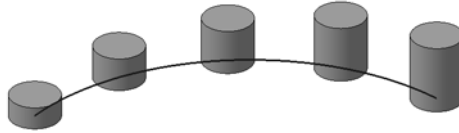


Fig. 2.7.7. Example of changing the dimensions of instances of the extrusion operation's array: new extrusion distance is set for each instance

Aside from the variables of an object being copied, the variables of the following objects used in the design can also be edited:

- ▼ sketches (if there are any dimensions with variables in them),
- ▼ auxiliary planes,
- ▼ curves (except splines and polylines),
- ▼ points.

For example, you can copy an extrusion item with a sketch built on a plane parallel to the XY plane and the extrusion depth is set according to the spacial position of the point. For copies of the array, you will be able to set new values ??of the variables that determine the dimensions of a sketch, the distance from its plane to the XY plane and the coordinates of the point to which you are extruding.

Afterwards, the variables of the copied objects include both these variables and the variables of objects used for building the copied object. Afterwards, the variables of an instance include a set of variables similar to those in the copied object.

You can manage the instance variables in two ways:

- ▼ using the table of variables (see Section 2.7.2.2),
- ▼ by editing the values of the instance dimensions (see Section 2.7.2.4 on p. 536).



You can also change the values of the instance variables after creating an array. To do this, you can use the Variables window (see Section 7.1.1.3 on p. 1571) or derived dimensions (see Section 2.9.2.8 on p. 648).

You **cannot** manage the instance variables in the following objects:

- ▼ in mirror arrays and arrays by sample,
- ▼ in other arrays:
 - ▼ when the bodies or surfaces are being copied,
 - ▼ when operations with the **Geometrical array** option enabled are being copied.

In all other cases, you **can** manage the instance variables. However, you cannot change variables of some types of objects. These include: sweep operation, operation by sections, operation of adding thickness, array, group of curves, group of points, and more. Copies of these objects in an array are always the same as in the original.

2.7.2.2. Table of modifiable variables

2.7.2.2.1. General Information

The **Table of modifiable variables** is a table saved in the model and containing variables of array instances.

The table of modifiable variables makes it possible to assign new values to variables of instances, thus modifying the instances' form and dimensions.

The table is formed by the user during array creation or editing. A table can be edited or deleted from the file in the future.

In order to change the form and dimensions of instances, create the table of modifiable variables so that it includes the required variables of instances. Then, assign new values to these variables.

New values of variables can be specified using two methods:

- ▼ by entering the new value in the respective cell of the table;
- ▼ by specifying the variables' increment values.

Array instances will be rebuilt in accordance to the new values of variables.

Creating the table of modifiable variables is described in Section 2.7.2.2.2 on p. 521, methods of assigning new values to variables are described in Section 2.7.2.2.4 on p. 526.

The table of modifiable variables saved in the model can be saved in the **data file** (see Appendix VI for more information about the data file). The table can also be read from the data file.



When creating an array by table, the table of modifiable variables can be used for setting instances' positions. Arrays by table are detailed in Chapter 2.7.7 on p. 565.

2.7.2.2.2. Creating, Editing and Deleting the Table of Modifiable Variables

Creation of the table of modifiable variables

To create the report on array, complete the following steps.

At Operation creation:



1. When creating or editing an array, click the **Table of modifiable variables** button on the Special Control Panel.

The following will be displayed:

- ▼ the **Table of modifiable variables** window with the table containing a column with instances' numbers;
- ▼ dimensions matching the variables of the object being copied (Fig. 2.7.8).

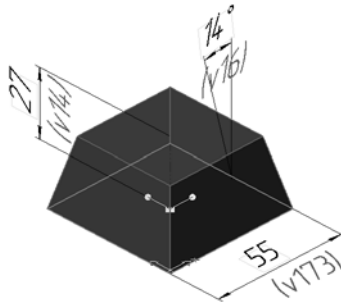


Fig. 2.7.8. Example of displaying dimensions corresponding to the extrusion operation variables.



If names of variables are not displayed on dimension lines, you can enable displaying them in the model window (see Section 2.9.2.2.3 on p. 637).



In case of the *array by table*, the table of modifiable variables contains coordinates of points that define positions of instances (see Section 2.7.7.2 on p. 568). You can enable/disable displaying coordinates in the table by clicking the **Define reference point coordinates** on the toolbar of the **Table of modifiable variables** window.

2. Create a set of variables of the object being copied, the values of which should be changed for instances.



Variables of some operations (*Hole from Library*, *Add Thickness*, *Cinematic Operation* and some others) cannot be added to the set.

For this purpose, specify the required dimensions of the object being copied in the model window. They are highlighted in color.

Columns corresponding to selected variables of the object being copied appear in the table. The first row contains the names of variables (headers of columns). In the second row, values of variables of the base instance are displayed. Each row of the table, starting from the third one contains the instance's number and empty cells.



An empty cell of the table of modifiable variables means that the variable in the cell has the same value as the value of the respective variable of the base instance. Therefore, the third and all other cells of the respective column will be empty immediately after adding the variable to the table.

Reselecting a dimension in the model window cancels its selection (the column related to the selected variable will be removed from the table).

3. Assign new values to variables. You can do this by the following methods:
 - ▼ by entering the new value of the variable in the respective cell of the table;
 - ▼ by specifying the variables' increment values.



Note that values of base instance's variables cannot be changed (the row with the values of the base instance is gray).

See section 2.7.2.2.4 on p. 526 for details about assigning new values to variables.

4. Click **OK** to close the **Table of modifiable variables** window.

The created table will be added to the file of the current model. Array instances will be rebuilt in accordance to the new values of variables.

See Section 2.7.2.2.3 on p. 524 for details about working with the table of modifiable variables.

When copying properties:



To create a table of modifiable variables, the component being copied must have external variables (for details about external variables, see Section 7.1.5.3 on p. 1593).



1. When creating or editing an array, click the **Table of modifiable variables** button on the Special Control Panel.

The **Table of modifiable variables** window with the table containing a column with instances' numbers will appear.



2. Create a set of variables of the object being copied, the values of which should be changed for instances. To do this use the **Read variables** or **Insert a variable** button in the **Table of modifiable variables** window. Description of button can be found in Section 2.7.2.2.3 on p. 524.
3. Perform steps 3 and 4 described above to create a table of modifiable variables when copying operations.



A table of modifiable variables is created automatically when you edit the dimensions of the array instances in the model window (see Section 2.7.2.4 on p. 536) or when you edit the variables matching these dimensions in the Variables window. The columns with modified variables are added into the table.

Editing the table of modifiable variables

Editing the table of modifiable variables is practically identical to its creating.



In order to begin editing the table, click the **Table of modifiable variables** button on the Special Control Panel.

Deletion of the table of modifiable variables



The table of modifiable variables may be deleted if it is no longer required. To do this, click the **Delete table** button on the Special Control Panel. As a result:

- ▼ modified values of variables for instances will be deleted from the file;
- ▼ instances' variables will accept the same values as the values of related variables of the base instance;
- ▼ instances will be rebuilt to become exact copies of the array's source object.

2.7.2.2.3. Working with the table of modified variables

The following options are available when working with the table.

Reading data from a previously created file



Click the **Read from file** button. Select the required file and press the **Open** button in the open file dialog. The table will be filled with data from the file (see Appendix VI for more information about the data file).

Saving the table to file



Click the **Save to file** button. The file save dialog will appear. Specify the file's location and name (by default, the file will have the *txt* extension) and click **Save**.

- ▼ If the table is saved during entry of variables' values, only numeric values of the variables will be saved in the data file. In case of array by table, coordinates of points defining instances positions can also be saved in the data file (see Section *Saving a table to data file* on p. 574 for more information about saving coordinates of points).
- ▼ If the table is saved during entry of variables' increment, only value steps will be saved in the data file.



When working with arrays by grid, by circular grid, along curve, or by points, **Read from file** and **Save to file** buttons will not be available if the table of modified variables is empty (has no columns with instances' variables).

When working with the array by table, these buttons are always available.

Adding variables of the copied object to the table



Click the **Read variables** button. Variables of the object being copied, that are not yet present in the table, will be added to it. Note that informative variables are not added to the table (see Section 7.1.1.2.1 on p. 1570 for more information about informative variables).

Adding a certain variable of the copied object to the table



Click the **Insert variable** button. The **Variables** dialog containing the list of all variables of the copied object appears on the screen. Specify the required variable and click the **OK** button. The selected variable will be added to the table.

Deletion of the selected column



Select the required column or any of its cells and click the **Delete column** button.

Changing the order of columns

Drag the column's header to the place you need (see Fig. 2.7.9).

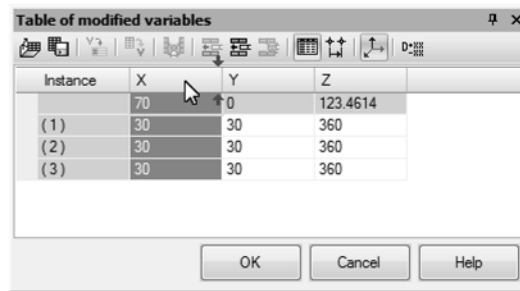


Fig. 2.7.9. Changing the order of columns in the Table of modified variables

Changing the number of rows in the table (for array by table)

Click the respective button:



- ▼ **Insert row above,**
- ▼ **Insert row below,**
- ▼ **Delete rows** (available only after selecting a row or a cell).

Changing the method of assigning values to variables for instances



- ▼ In order to use the method that makes it possible to enter arbitrary values in the table's cells, click the **Values of Variables** button.
- ▼ In order to use the method that makes it possible to specify the change value step, click the **Value step** button.

Details about assigning new values to variables for instances can be found in Section 2.7.2.2.4 on p. 526.

If a column contains values that do not form arithmetical progression (i.e. value step cannot be defined), entering the mode for setting the value step is only possible if data is deleted from this column. After clicking the **Value step** button, the system will request setting base values.

- ▼ Click **Yes** to delete the value in the column and begin setting the value step.
- ▼ Click **No** to retain the value in the column. In this case, the system will not enter the mode for value step setting.

Enabling/disabling display of coordinates in the table (for array by table)



Click the **Specify base point coordinates** button.



If an array of arbitrary objects is created (not the objects of a certain type), the **Specify base point coordinates** button will not be available in the toolbar.

Cancel changes in the table

Cancel changes in the table means deletion of new values from its cells. Empty table cell means that the value in it is the same as the base one.

▼ Cancel changes in cells

Click the required cell in the table and call the **Base Value** command from the context menu.

▼ Cancel changes in columns

Click the header of the required column and call the **Base Value** command from the context menu.

▼ Cancel all changes in the table

Call the **All base values** command from the context menu of the **Table of modified variables** window.



Enabling/disabling display of individual tolerances in the table

Click the **Display individual tolerances** button. More information about individual tolerances in the table can be found in Section 2.7.2.2.5 on p. 529.

2.7.2.2.4. Changing the value of an instance's variable

New values of instances' variables can be specified using two methods:

Method 1



Entry of the variable's value into the cell of the table of modifiable variables.

In order to use this method, click the **Values of Variables** button in the **Table of modifiable variables** window.

A table with the following structure will appear in the window (Fig. 2.7.10):

- ▼ The first column contains numbers of the array's instances.
- ▼ The first row contains the names of variables of the object being copied (headers of the table's columns).
- ▼ In the second row, values of variables of the base instance are displayed. Values in the row are not available for editing (the row is gray).
- ▼ Each row of the table, starting from the third one contains the instance's number and a set of variables modified by the user. An empty cell in the table means that the variable in the cell remains unchanged, i.e. its value is the same as the value of the respective variable of the base instance.

Instance	v9	v10	v14	v16
(1.1)	20	5	25	5°
(2.1)	25			6°
(3.1)				7°
(1.2)	30	3	30	6°
(2.2)				6°
(3.2)				7°
(1.3)	40	4	40	7°
(2.3)	40	4	60	7°
(3.3)				7°

Fig. 2.7.10. Example of the table of modifiable variables for entry of a new variable's value in the table's cell (array by grid)

To assign a value to a variable, you can enter a number or a mathematical expression into a cell in the table. Further steps are identical to those described in Section 7.1.3.3 on p. 1579.

Activate the required cell by double-clicking it, enter the new variable's value in it and click <Enter>. To move between cells, use arrow keys.

Individual tolerances may be assigned to dimensions of the sketch or operation of the object being copied (see Chapter 2.10.2 on p. 667). If the table contains variables of such dimensions, you can enable displaying tolerances in the table. For this purpose, the **Display individual tolerances** button is used (see Fig. 2.7.10 on p. 527). When working in the table, you can change tolerances for dimensions of the sketch or operation (see Section 2.7.2.2.5 on p. 529).



Method 2



Specifying the variables' increment value.

In order to use this method, click the **Increment value** button in the **Table of modifiable variables** window.

A table with the following structure will appear in the window (Fig. 2.7.11):

- ▼ The first row contains the names of variables of the base instance (headers of the table's columns).
- ▼ In the second row, values of variables of the base instance are displayed. Values in the row cannot be edited.
- ▼ The third row is used for setting the increment value for variables.

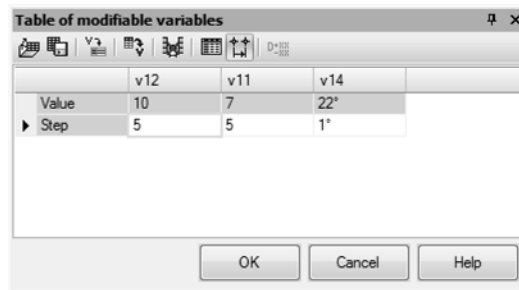


Fig. 2.7.11. Example of the table of modifiable variables for specifying the variables' increment value

In the third row of the table, activate the required cell by double-clicking it, enter the increment value in it and click <Enter>. The step value can be both positive and negative. To move between cells, use arrow keys.



If necessary, you can view the new values of variables. For this purpose, click the **Values of Variables** button in the **Table of modifiable variables** window. The table with the variables' values will be displayed in the window. The column of the variable, for which increment value was assigned, will contain its new values.

Note that the value of the variable that will be assigned to a certain instance depends on its **number and array type**.

- ▼ In arrays by parallelogram or circular grid, values of variables for instances are changed in accordance with the diagrams shown in Fig. 2.7.12 and 2.7.13.

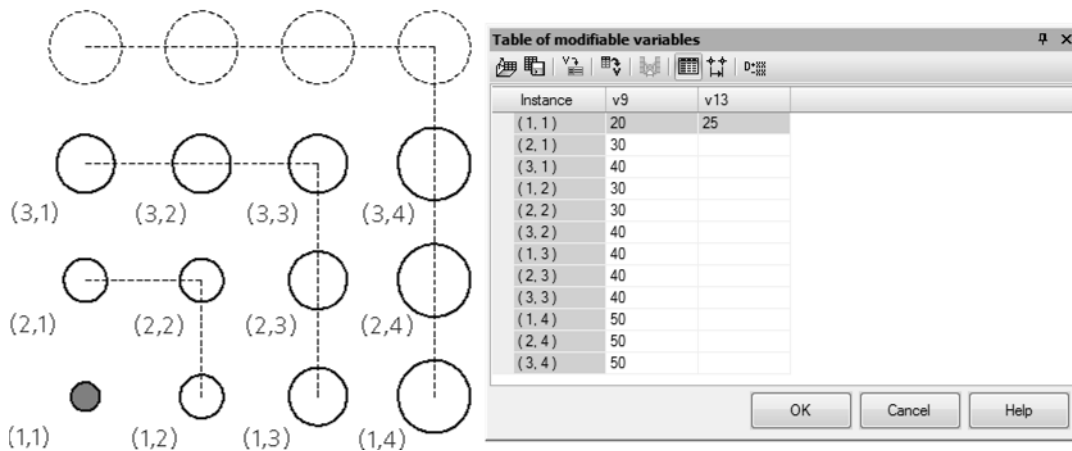
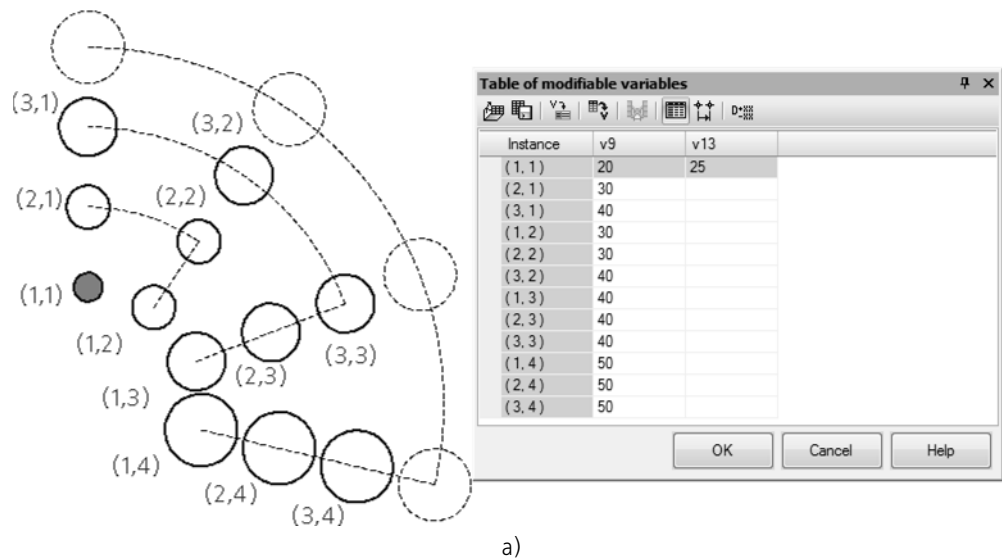


Fig. 2.7.12. The diagram of changing variables' values with a specified increment for instances of an array by parallelogram grid



a)

Fig. 2.7.13. The diagram of changing variables' values with a specified increment for instances of an array by circular grid

- ▼ In arrays by curve, by points, and by table, values of variables for instances are incremented along the instances' numbers (Fig. 2.7.14).

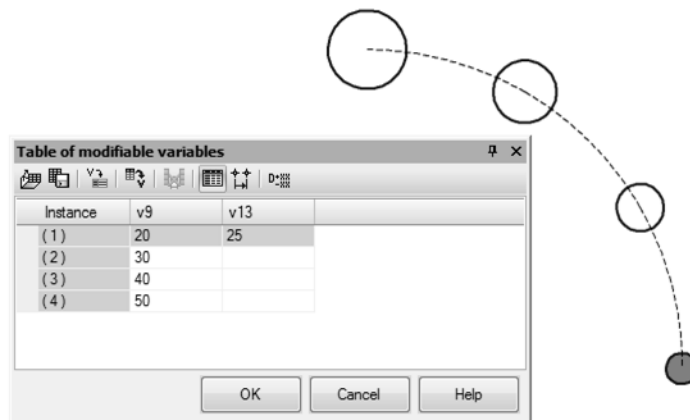


Fig. 2.7.14. The diagram of changing variables' values with a specified increment by the order of instances' numbering (creation of an array by a curve)

Thus, in the first case, some instances have similar values of the variable (Fig. 2.7.12 and 2.7.13); in the second case, all variables' values will be different (Fig. 2.7.14).

2.7.2.2.5. Individual tolerances in the table of modifiable variables

You can assign tolerances (general and individual, see Chapter 2.10.2 on p. 667) to values of instances' parameters (guiding dimensions of the sketch and operation's parameters). If indi-



vidual tolerances are assigned, you can enable displaying them in the table. For this purpose, the **Display individual tolerances** button is used (see Fig. 2.7.15 on p. 530).



You can view individual tolerances in the table of modifiable variables, if new values of instances' parameters are set using a method supporting manual entry of arbitrary values in the table's cells (**Method 1**, see 2.7.2.2.4 on p. 526).

If the **Display individual tolerances** button is enabled, the following data is displayed in the table's cells:

- ▼ The cells of the second table's row contain the base instance's data;
 - ▼ values of the base instance's parameters,
 - ▼ assigned tolerance – class of accuracy and maximum deviations or limit values of the base instance parameters.
- ▼ In other rows of the table starting from the third one, the following information is displayed:
 - ▼ a set of modified values of instances' parameters,
 - ▼ tolerances for the values of instances' parameters.

Instance	v12	v113	v14
(1. 1)	59h14(-0,74)	27	14'(+1';-1')
(2. 1)	70h8(-0,046)		15'(+1';-1')
(3. 1)		90h14(-0,87)	16'(+1';-1')
(4. 1)			17'(+1';-1')

Fig. 2.7.15. Displaying individual tolerances in the table of modifiable variables

Note the following specifics of object property displaying in a drawing.

- ▼ An empty cell of the table means that the value and tolerance of the instance's parameter in this cell are the same as the value and tolerance of the respective parameter of the base instance.
- ▼ A cell contains only the value of the parameter — this means that either general tolerance or no tolerance is assigned to the instance parameter in this cell.
- ▼ A cell contains both the value of the parameter and the specified tolerance — this means that the value and/or tolerance of the instance in this cell differ from the value and/or tolerance of the corresponding parameter of the base instance.



If the **Display individual tolerances** button is enabled, cells of the table of modifiable variables are not available for manual entry.

In the table of modifiable variables, you can assign or change tolerances for instances' parameter values. For this purpose, the **Value and tolerance** dialog should be used.

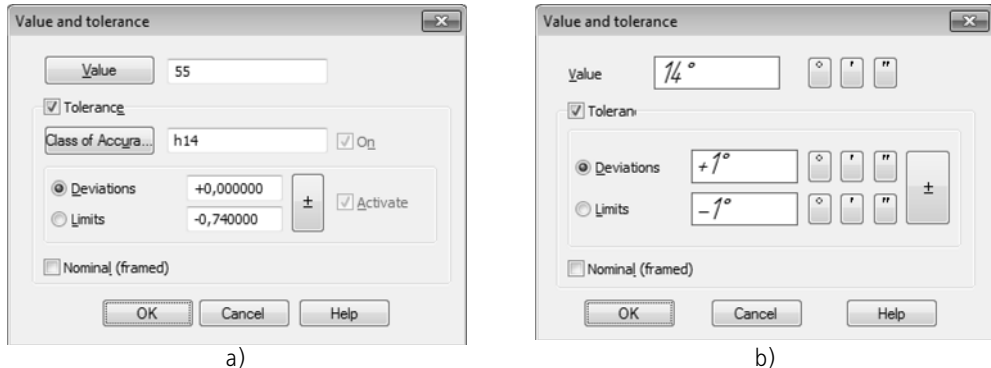


Fig. 2.7.16. The **Value and tolerance** dialog
a) for a linear value, b) for an angular value

The dialog is called using the following methods:

- ▼ by the **Assign tolerance** command of the table cell's context menu.
- ▼ if **Display individual tolerance** button is enabled, by clicking the ellipses button in the right part of the cell, or by double-clicking the cell.

Controls of the **Tolerance** group are described in Table 3.3.5 on p. 982. Assigning tolerance procedure is the same as the procedure described in Section 2.10.2.2 on p. 669.

The dialog also makes it possible to assign a new value of the instance's parameter. For this purpose, the **Value** field and the respective button (the button is present in the dialog for instances' parameters whose values are expressed in linear values) are used. You can input a new value using the keyboard. For an instance expressed in linear values, you can also select a value from the user menu. To do so, click **Value**. The user menu will appear on the screen. By default, it contains normal linear dimensions by GOST 6636-69.

After performing all necessary operations in the dialog, close it by clicking the **OK** button.



When the value of the variable is a mathematical expression, you cannot change the parameter value in the tolerance configuration dialog. The **Value** field and a same-name button are not displayed on the dialog.



You cannot change the data of the base instance (the table row containing the values of the base instance is grayed out).

2.7.2.3. Report on array with the table of modifiable variables

If an array contains the table of modifiable variables (see Section 2.7.2 on p. 519), you can create a report on the array. The report is a table with information from the table of modifiable variables (a set of variables and their values, point coordinates). The array report's table can be placed in a graphical or text document. You can edit data in the table, change its header, location and so on.

The report on the array can be linked associatively to the source of data (a model) or not linked to it. The associative link is formed during creation of the report. The procedure of creating the report on array can be found in Section 2.7.2.3.1 on p. 532.

If the report on array is associatively linked to the source of data, you can update data in the report, open the source document and use commands for editing that are available for associative reports (see Section 5.2.3.8 on p. 1433). For a report on array which is not linked to a data source, these options are not available.

You can work with the table of the report on array in the same way as when working with the table of variants (see Section 2.13.4.3 on p. 776).

Parameters of the table of the report on array are arranged and set up during its creation or editing (see Sections 2.7.2.3.2 on p. 533 and 2.7.2.3.4 on p. 535). Text of a table located in a graphical or text document is displayed in accordance with the setup of text parameters of the report's table applied to that document (see Sections 9.2.4.7 on p. 1754 and 9.2.6.23 on p. 1835).

2.7.2.3.1. Creation of the report on array with the table of modifiable variables

In order to create the report on array with the table of modifiable variables, do the following:

1. Open a model containing the array with the table of modifiable variables (see Section 2.7.2.2 on p. 521 for details about the table).
2. Call the **Create report on array with the table of modifiable variables** from the **Service – Reports** menu or using the respective button on the **Reports** bar.



Elements for creation of the report will appear in the Property Bar (Fig. 2.7.17).

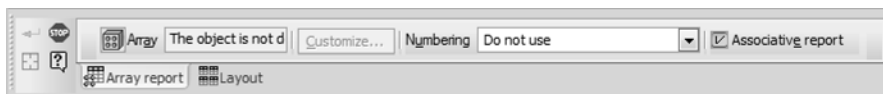


Fig. 2.7.17. Property Bar during creation of the report on array

3. Select the required array in the Design Tree or in the model window.



It is impossible to select an array without the table of modifiable variables.



The selected array will be highlighted. The name of the selected array is displayed in the field beside the **Array** switch.

After selection of an array, the setup dialog for the table of the report on array appears on the screen (see Fig. 2.7.18 on p. 534).



In order to reselect an array for creating the report, close the setup dialog of the report table and select the required array. Once the array is selected, the setup dialog for the table of the report will appear on the screen again.



The array, for which the report should be created, can be selected before calling the command. In this case, the setup dialog for the table of the report should be called by clicking the **Setup...** button on the Property Bar.

4. Set up parameters of the table of the report on array (see Section 2.7.2.3.2 on p. 533).
5. Using the **Associative report** option, enable/disable creation of the associative link between the report being created and the source document (model).
 - ▼ If the option is enabled, the created report will be linked associatively to the source document. Changes in the source document (in the table of modifiable variables) will be transferred to the report.
 - ▼ If the option is disabled, then the report is created without the associative link to the source document. In this case, changes in the source document will not be transferred to the created report.



The **Associative report** option is available on the Property Bar during creation of the report on array as long as the source document is saved to a file on the disk. During the report editing, the option is present on the Property Bar.



The associative report on array should be placed in a graphical document. If placed in a text document, the associative link between the report and the source document will not be created.

6. You can enable automatic assignment of numbers to report tables. To do this, choose the numbering group from the **Numbering** drop-down list (for details about numbering, see 3.9.1 on p. 1277).
7. Set up the table contents for the report on array (see Section 2.7.2.3.4 on p. 535).



8. In order to finish creating the report, click the **Create Object** button.

This will display the dialog to select a document or the standard dialog to open files. Open the required graphical or text file.

 - ▼ Locate the ghost of the report table in the graphical document. For this purpose, specify the location of the table's snap point by the cursor or by entering coordinates on the Property Bar and click the left mouse button to fix the table. If necessary, you can modify it (see Section 4.2.2.1.3 on p. 1364).
 - ▼ In a text document, the report table is located automatically.

2.7.2.3.2. Setup of the table of the report on array

Setup of the table of the report on array is performed in the **Parameters of the table of the report on array** dialog (Fig. 2.7.18).

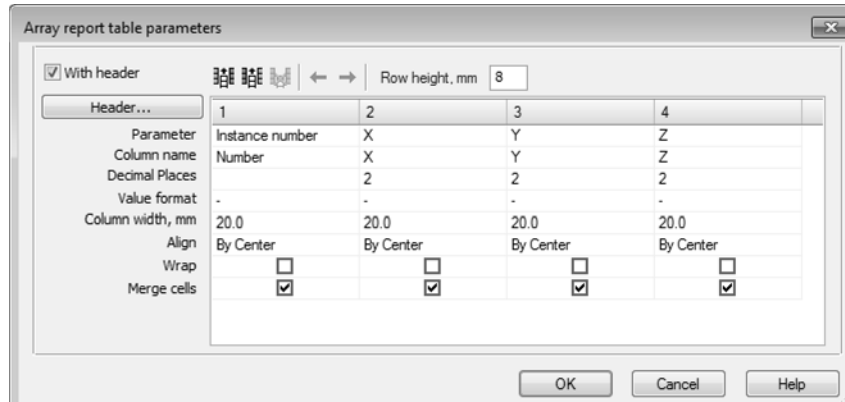


Fig. 2.7.18. Setup dialog for the table of the report on array



The dialog is called during creation or editing of the table of the report on array:

- ▼ automatically after selection of an array;
- ▼ or by clicking the **Setup...** button on the **Report on array** tab of the Property bar.

By default, the dialog shows the columns matching the columns in the table of modifiable variables of the selected array, as well as a column containing the numbering of the instances. You can remove or restore any columns in the report.



Pay attention to the following feature of the table of modifiable variables in the array by table. In this array, the table of modifiable variables always contain the columns of coordinates of the array instances, but you can show or hide them as required. The same columns in the report's table can be either included or excluded depending on whether they are currently enabled in the array.

In the **Parameters of the table in the report by array** dialog, you can do the following.

- ▼ Selection of the table display option – with header or without it. For this purpose, the **With header** option is used.
- ▼ Adding, deletion, and moving of the table's columns; controls are the same as those used for the report table setup (see Table 5.2.2 on p. 1408).
- ▼ Setup of the table's columns display parameters and data contained in them (see Section 2.7.2.3.3 on p. 535).
- ▼ Setting the table's row height. For this purpose, the **Row height, mm** field is used. The value is manually entered in this field.
- ▼ Setup of the table's heading. The table's heading is set up in the same way as the heading of the table of variants (see Section 5.2.2.2.4 on p. 1411).

In order to finish the table setup, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.



Columns of the table of the report on array depend on the table's settings. In order to change the set of columns of the table of the report on array, it is necessary to change its settings. Changing the set of columns of the modifiable variables table (in the source document) is not transferred to the associative report on array.

2.7.2.3.3. Display setup for columns of the table of the report on array

The following parameters can be set up for table columns of the report on array:

- ▼ **Column name, Decimal places, Column width, mm, Alignment, Wrap**; the setup is the same as for the report style parameters described in Table 5.2.3 on p. 1410,
- ▼ **Parameter, Value format, Merge cells**; the setup is described in Table 2.7.2.

Tab. 2.7.2. Display setup parameters for columns of the table of the report on array

Element	Description
Parameter	This field* contains the name of the array instances' parameter (for example, coordinates of points that define instances' positions, names of variables). Choose the parameter's name from the list. The list of instances' parameters corresponds to columns of the table of modifiable variables in the selected array.
Value format	This field* makes it possible to choose the variant of displaying the value of the instance's parameter in the table: <ul style="list-style-type: none"> ▼ <i>Value</i>, ▼ <i>Value + tolerance</i>, ▼ <i>Tolerance</i>.
Merge cells	This option makes it possible to merge adjacent cells in a column of the table, if the cells' contents is identical, i.e. if the displayed value is the same for these adjacent cells.

* The control allows to select an option from the drop-down list. After you click a cell, in its right corner there appears a button with a triangle. Press the button to display the list for selecting a parameter option.

2.7.2.3.4. Array report table contents

Information about parameters of the array's instances can be presented in more than one table. The setting that is used to select the method of positioning tables on a sheet is made on the **Contents** tab of the Properties Bar (Fig. 2.7.19).

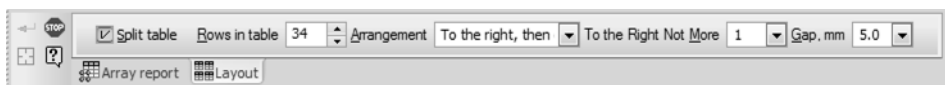


Fig. 2.7.19. **Contents** tab of the Properties Bar

In order to create several tables and adjust their positions on the sheet, perform the following actions.

1. You can have several tables by enabling the **Split table** option.
2. In the **Number of rows** in table, specify the maximum number of rows in each table. Display parameters are the same for all tables.
3. To specify the order of tables' positioning on the sheet, choose the required direction of positioning from the **Contents** drop-down list.
- ▼ The **To the right, then downwards** option positions tables in horizontal rows. The number of tables in a row is specified in the **To the right, not more than** field.
- ▼ The **Downwards, then to the right** option makes it possible to position tables in vertical rows. The number of tables in a row is specified in the **Downwards, not more than** field.
4. Enter the value of distance between tables in the **Gap, mm** field. You can enter the required value of the distance manually or select it from the drop-down list.

2.7.2.4. Dimensions of the array instances

When working with all types of arrays, except the mirror arrays and arrays by sample, you can adjust the shape and dimensions of the instances by changing the **dimensions of sketches and operations** of these instances.

The dimensions of the sketches and instances correspond to the same dimensions of the object being copied.

To display the dimensions of the sketches and operations for a certain instance in the model window, perform the following.

1. When creating or editing an array, expand the **Instances** tab on the Properties panel.
2. In the **Mode** group, enable the **Edit dimensions** switch.
3. Choose the instance you want on the **List of instances** panel.
4. On the **List of objects** panel, choose the object (a sketch or an operation) which dimensions you want to display. The dimensions of the object will appear in the model window.



Dimensions of sketches and operations are not displayed in the Design Tree.

You can edit values of the following dimensions: This can modify the geometry of the array instance. The appearance and methods for the dimensions of the sketches and operations of the array instances are similar to those described in Section 2.9.2.2 on p. 632.



When a dimension is changed, its variable is added to the table of modifiable variables of the array (if this table does not exist, the system will create it).

For arrays by table, you can also adjust the position of the instances by changing the values of **instance position dimensions**.

These dimensions are created when you set the coordinates of points which define the positions of the array instance, and they match these coordinates. The variables of the position dimensions are added into the Variables window.

By default, the dimensions of the instance positions are not shown in the Model window. To display the dimensions of the position for a certain instance, choose it on the **List of instances** panel (see step 3 described above).

To change the position of an instance, edit the values of dimensions. You can work with position dimensions similar to the dimensions of sketches and operations.



If the coordinates of points which define the positions of the array instance are taken from a file, then the values of the position dimensions cannot be edited. In this case, the dimensions of positions are not displayed.

The tolerances for dimensions of the sketches and operations of array instances are the same as for the corresponding dimensions of an object being copied. By default, the dimensions of positions are assigned general tolerances.

When editing a value of a dimension, you can assign it another tolerance or even remove the tolerance at all.



You can also adjust the geometry and positions of the array instances when required using the derived dimensions 2.9.2.8 on p. 648. These dimensions can be created for the entire array, for an array instance, or for objects included into an array instance.

The dimensions are created after creation of the array.

2.7.3. By grid array

You can create an array of objects by positioning them in parallelogram grid nodes, for example, as is shown on the Fig. 2.7.20.

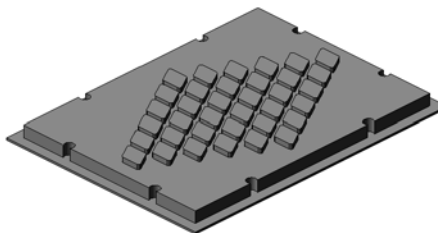


Fig. 2.7.20. Example of array along grid

The grid formation scheme is described in Section 2.7.3.1, array construction procedure — in section 2.7.3.2.

2.7.3.1. Formation scheme of parallelogram grid

Parallelogram grid is characterized by the direction of its generating vectors and the angle between them (Fig. 2.7.21). Any point of copied objects may be the origin of the grid coordinates.

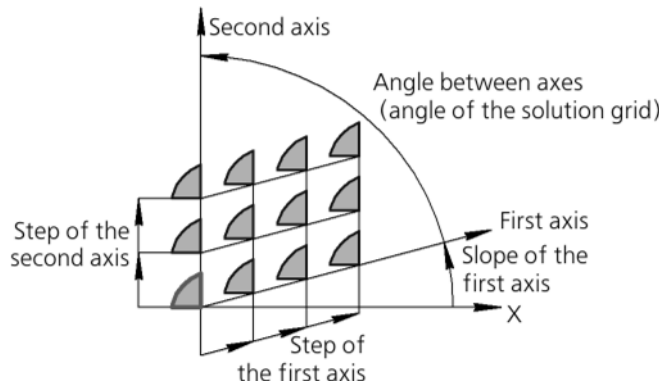


Fig. 2.7.21. Formation scheme of parallelogram grid

Array instances are displayed as ghosts and positioned in space in accordance with the preset parameters of the grid. A ghost allows a visual control over parameter accuracy.

2.7.3.2. The Procedure for Construction of a Array by Grid

To create an array by grid, complete the following steps.



1. Call the **Array by Grid** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).

2. Specify the **copied object**: operations or bodies or surfaces or curves, points, auxiliary planes and axes, or components.

For details about copied object selection, see Section 2.7.1.3.1 on p. 512.

Names of the selected objects are displayed on the **List of objects** bar; the array's ghost will appear on the screen.



You can also select objects for copying prior to calling the array creation command.

Once objects for copying have been selected, **Parameters**, **Deleted**, **Properties** tabs will appear on the property bar.

3. Go to the **Parameters** tab.
 - 3.1. Using the tab's control elements, set **grid parameters** for the first axis (see Section 2.7.3.3 on p. 539), then for the second axis (see Section 2.7.3.4.1 on p. 540).
To set the array grid parameters, you can use hot points (see Section 2.1.4.2.2 on p. 138 for description of hot points).



You can arrange the array grid parameters using the hot points without switching to the **Parameters** tab.

All the changes will be reflected in the array ghost.

4. If required, you can also:
 - ▼ Changing array instances' positions with respect to the **base instance**— the instance which corresponds to the objects to be copied (see Section 2.7.3.5 on p. 541).
 - ▼ Selection of **copy positioning method** inside the grid (see Section 2.7.3.6 on p. 542).
 - ▼ Creating a **geometric array** with operations to be copied (see Section 2.7.1.3.5 on p. 515).
 - ▼ Deleting and restoring array instances. Enable the **Instances** tab of the Properties Bar. More details about deleting or restoring instances can be found in Section 2.7.1.3.6 on p. 516.
 - ▼ Managing form and dimensions of instances by changing their variables' values (see Section 2.7.2 on p. 519).
 - ▼ Changing of array name and properties of surface of instances. Enable the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.

2.7.3.3. Grid parameters along the first axis

2.7.3.3.1. First Axis Direction

When copying an operation, direction of the first grid axis by default coincides with the *X* axis of the sketch for this operation.

If you are copying extrusion or rotation operations that use other objects – not a sketch (for example, a face) – as a section, the direction of the first grid axis by default will coincide with *X* axis of the absolute coordinate system.



If several operations are selected for copying, the direction of the first grid axis is set by *X* axis of the sketch or by the *X* axis of the absolute coordinate system of the first specified operation.

When copying other types of objects, the direction of the first default grid axis coincides with *X* axis of the current coordinate system (see Section 2.8.3.3 on p. 597 for details on selection of the current coordinate system).

The name of the guide object for the first axis will appear in the **Axis 1** field on Property bar.



You can change the first axis direction as follows:

1. Set angle between the first axis and *X* axis of the current coordinate system in *XY* plane or *X* axis in the operation's sketch plane. To do this, enter the value of the first grid axis slope in the **Slope** field.
2. Select the guide object in the window or in the Design Tree. You can use any rectangular or flat object as a guide, as well as a cylindric or conical surface. Straight and flat objects are listed in Table 2.1.7 on p. 131.

Direction, defined by a straight object — a line parallel to the object.

Direction, defined by a plane object — a line perpendicular to the object.

The direction defined by a cylindrical or conical surface is a line parallel to the rotation axis.



The specified object's name will appear in the **Axis 1** field.

3. Construct a vector specifying the axis direction. In this case, the word “Vector” will be displayed in the **Axis 1** field. The vector can be created with the **Vector Drawing** button on the Special Control Panel. For more information on drawing a vector, see Chapter 2.8.5 on p. 613.



The resulting axis direction is the **Forward Direction**. To reverse the axis direction, activate the **Reverse Direction** switch in the **Direction** group.





2.7.3.3.2. Grid Step along First Axis

To specify grid step along the first axis, complete the following steps.

1. Enter the number of copies in the direction of the first axis into field **N 1**.
2. Enter the value of step between copies into the **Step 1** field.
3. Activate the switch matching the specified value of step in the **Mode 1** group (see Table 2.7.3).

Tab. 2.7.3. Mode to specify a grid step

	Value of Mode option	Method to specify a step
	Step between adjacent instances	The system regards step value as the distance between the corresponding points of neighboring copies along the axis direction [*] . This option is convenient for placing a certain number of copies at a definite distance from each other.
	Step between boundary instances	The system regards step value as the distance between corresponding points of the first and the last copies along the axis direction [*] . This option is convenient for placing a certain number of copies on a line of the specific length.

^{*} If the number of copies along the axis direction is equal to two, then the result of the construction shall be the same for any position of the **Mode** switch.



The same results of construction an array can be reached under various positions of the **Mode** switch. To do this, you should take into account the following correlation:

$$L = (N - 1) \cdot I, \text{ where}$$

L is a step between outer instances,

N is the number of copies

I is the step between adjacent instances.

2.7.3.4. Grid parameters along the second axis

2.7.3.4.1. The second axis direction



To specify the second grid axis direction, enable the **Axis 2** switch on the property bar.

The second axis direction controls will be displayed on the Property bar.

You can specify the second axis direction as follows.

1. In the **Opening Angle** field enter the value of the angle between the grid axes;
2. Specify a guide object in the Design Tree or in the model window (for information about the guide object, see p.2 of Section 2.7.3.3.1 on p. 539). The object name will appear in the **Axis 2** field.
3. To create a vector which defines axis direction (see p.3of Section 2.7.3.3.1 on p. 539). In this case, the word “Vector” will be displayed in the **Axis 2** field.



The resulting axis direction is the **Forward Direction**. To reverse the axis direction, activate the **Reverse Direction** switch in the **Direction** group.

2.7.3.4.2. Grid Step along the Second Axis

The grid step parameters along the second axis are the same as the step parameters along the first axis (see Section 2.7.3.3.2 on p. 540). To specify them, use fields and switches **N 2**, **Step 2** and **Mode 2**.

2.7.3.5. Changing positions of instances in an array against the base one

By default, the number of the base instance (the instance that corresponds to the object to be copied) is 1.1 (see Fig. 2.7.22, a). Array instances numbers are described in section 2.7.1.1.2 on p. 510.

You can change the array instances' position with respect to the base instance by assigning a certain number to it in the array (see Fig. 2.7.22, b).

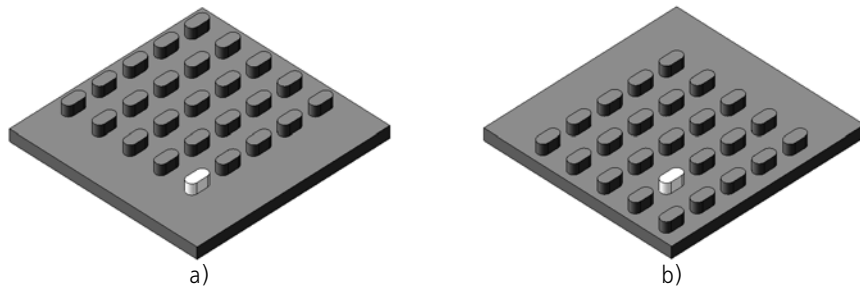



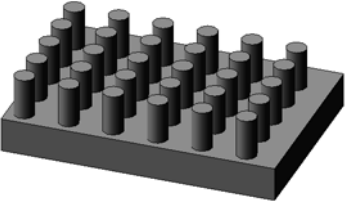

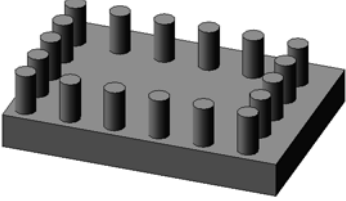

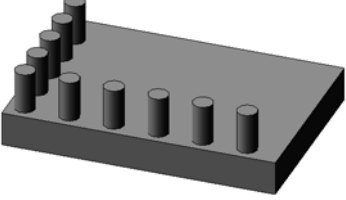

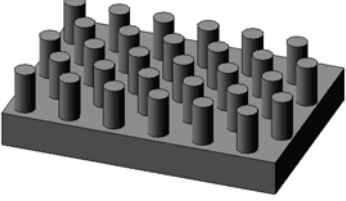
Fig. 2.7.22. Changing instances' position in an array against the base one (the base instance is highlighted in white color): a) base instance with number (1.1); b) base instance with number (2.2)

- ▼ To change the base instance's number in an array, enable the **Base instance** option on the property bar. Fields **Number 1** and **Number 2** will appear on the property bar. Enter or set with counter the required numbers of base instance by grid axes in these fields. The array's ghost will be re-build – the array's instances are positioned in such a way that the number of the base instance in the array corresponds to a given number.
- ▼ To restore the base instance's default number 1.1, disable the option. The array's ghost will be rebuilt– the instances will be positioned so that the base instance number is used by default.


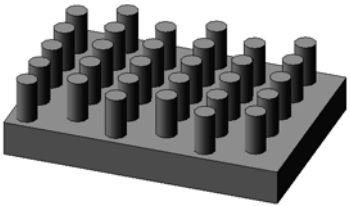
2.7.3.6. Positioning copies inside the grid

Positioning of array's instances and their number inside of the initially defined grid can be modified, for example, instances can be arranged along the grid's perimeter only or in chessboard fashion. In order to define a positioning variant, select a line from the **Scheme** list (Table 2.7.4).

Tab. 2.7.4. Control over copies inside the grid

	Positioning scheme	Procedure to create an array Construction result	
	Standard scheme	Array's instances are positioned in all nodes of the grid.	
	Delete Copies Inside Grid*	Array's instances are positioned along the perimeter of the grid.	
	Copy only along the axes**	Array's instances are positioned along the grid's axes only.	
	Staggered order — displacement along the first axis**	The instances are displaced with respect to nodes of the defined grid by half the step along the first axis. The displacement takes place in each even row of the current direction of the axis. The first row is the row containing the instance number (1, 1).	

Tab. 2.7.4. Control over copies inside the grid

	Positioning scheme	Procedure to create an array	Construction result
	Staggered order — displacement along the second axis**	The instances are displaced with respect to nodes of the defined grid by half the step along the second axis. The displacement takes place in each even row of the current direction of the axis. The first row is the row containing the instance number (1, 1).	

* The scheme makes sense, if the number of instances in both directions is three or more.

** The scheme makes sense, if the number of instances in both directions is two or more.



The position of the reference instance if positioned in chess-board fashion will not be changed.

2.7.4. Pattern by Circular Grid

You can create an array of objects by positioning them in concentric grid nodes, for example, as is shown on the Fig. 2.7.23.



Fig. 2.7.23. Example of array along circular grid

The grid formation scheme is described in Section 2.7.4.1 on p. 543, array construction procedure — in section 2.7.4.2 on p. 546.

2.7.4.1. Array construction scheme

The circular grid is characterized by the position of its plane and center, radii of circles and the angle between the respective radial rays (Fig. 2.7.24).

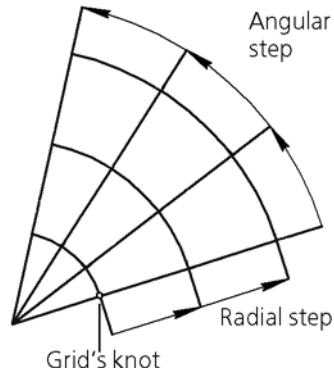


Fig. 2.7.24. Concentric grid of the array

The position of the grid's plane and its center is defined by the **array axis** and **base point of the base instance**.

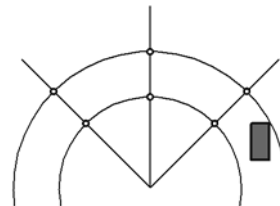
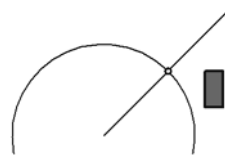
Base point position can be **autodefined** or specified **manually**.

Array creation by concentric grid is described in Table. 2.7.5.

Copying of a group of objects has the following peculiarities.

Tab. 2.7.5. Constructing Array of Components by Circular Grid

Construction Spacing	Result*
1. The grid's plane going through the base point of the base instance perpendicular to the array's axis will be built.	
2. In the grid's plane, the first ray and the first circle of the grid going through the base point will be built. Thus, the base point of the base instance will become the first node of the grid.	
3. The remaining rays and circles of the grid are built and other nodes of the grid are determined in accordance with the grid's parameters.	

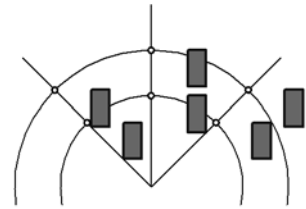


Tab. 2.7.5. Constructing Array of Components by Circular Grid

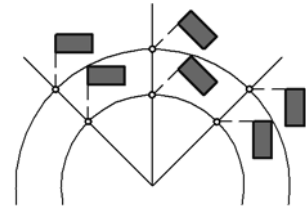
Construction Spacing

Result*

4. The base points of other array's instances are mated with the grid's nodes. The instances are positioned against their base points in the same way as the base instance is positioned against its base point.



5. If the Set Copies Radially option is enabled, each instance will be rotated in the grid's plane against its base point so that it is positioned against its ray the same way as the base instance is positioned against its base ray.



* The construction result is shown as a projection on the plane perpendicular to the array's axis.

If a group of objects is being copied, the array for all objects, except for operations, will be constructed as follows:

- ▼ If the base point is set manually, the group of objects will be copied as **a whole**: one grid is built for the group of objects to be copied; if additional turn is enabled, each instance (group of copies) will be rotated around its base point (Fig. 2.7.25, a).
- ▼ If a base point is autodefined, the group's objects will be copied separately: a separate grid will be created for each object; if additional turn is enabled, each object's copy will be rotated around its base point (Fig. 2.7.25,b).



The group of operations is always copied as a whole.

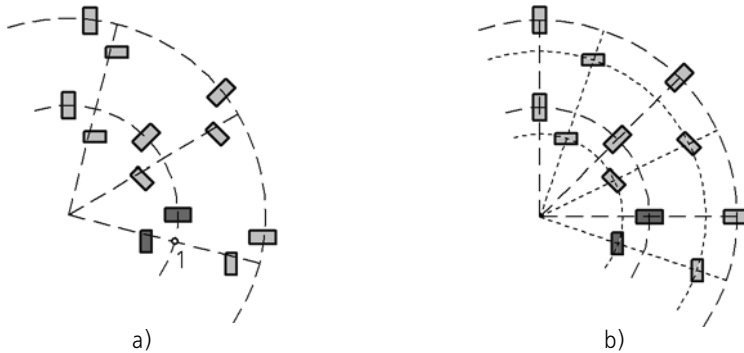


Fig. 2.7.25. Array of the group of objects along the concentric grid with additional turn:
a) point 1 is specified as the base point, b) base point is not specified

2.7.4.2. Constructing Array of Components by Circular Grid

To draw an array by circular grid, complete the following steps.



1. Call the **Array by Circular Grid** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).

2. Specify the **copied object**: operations or bodies or surfaces or curves, points, auxiliary planes and axes, or components.

For details about copied object selection, see Section 2.7.1.3.1 on p. 512.

The names of the selected objects will appear in the **List of Objects** toolbar.

The ghost of the **reference point of the object being copied** appears on the screen as three mutually perpendicular vectors (see Section 2.1.1.4 on p. 104).

Parameters, Deleted, Properties tabs will appear on the property bar.



You can also select objects for copying prior to calling the array creation command.

3. Go to the **Parameters** tab.

- 3.1. Specify the **array axis**. You can use any straight line object or revolution surface (except for sphere) as an array axis. Straight objects are listed in Table 2.1.7 on p. 131.



Enable the **Axis** switch on the property bar and specify the required object in the model window. Construction axes can also be selected in the Design Tree.

The object-axis name will appear in the **Axis** field.

Once the array's axis has been selected, the array's ghost appears on the screen together with arrows indicating the direction of adding instances with respect to copied elements. The grid shall be built with default settings. Using the ghost's hot points, you can change default parameters of the array's grid.



- 3.2. Enable the **Circular direction** switch in the **Direction** group. Specify **grid parameters in circular direction** (see Section 2.7.4.3 on p. 547).



3.3. Enable the **Radial direction** switch in the **Direction** group. Specify **grid parameters in radial direction** (see section 2.7.4.4 on p. 548).

3.4. Select **Orientation of Array Instances** (see Section 2.7.4.5 on p. 549).

All the changes will be reflected in the array ghost.

4. If required, you can also use the following options:

- ▼ Instance displacement along the **array's axis** (see Section 2.7.4.6 on p. 550).
- ▼ Changing array instances' positions with respect to the **base instance** — the instance which corresponds to the objects to be copied (see Section 2.7.4.7 on p. 551).
- ▼ Changing the position of the **base instance's reference point** (see Section 2.7.4.9 on p. 553).
- ▼ Disabling using the **projection of the reference point of the object being copied — available, if rotation bodies/surfaces are copied** (see Section 2.7.1.3.3 on p. 514).
- ▼ Creating a **geometric array** — available, if operations are copied (see Section 2.7.1.3.5 on p. 515).
- ▼ Deleting and restoring array instances. Enable the **Instances** tab of the Properties Bar. See Section 2.7.1.3.6 on p. 516 for information on the deletion and restoring array's instances.
- ▼ Managing form and dimensions of instances by changing their variables' values (see Section 2.7.2 on p. 519).
- ▼ Changing of array name and properties of surface of instances. Enable the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.



If base instance's reference point is on the array's axis, it is impossible to define the direction of the grid's first axis (and, consequently, of the other axes). The number of instances in the radial direction of such an array is always equal to 1. Changing this number, as well as other parameters of the grid in the radial direction, is not possible (the **In radial direction** switch is not available). Such an array is not defective. If it suits you, do not change its source objects. If it is necessary to change the grid's parameters in the radial direction, choose another array's axis or another base point.

2.7.4.3. Grid Parameters in Circular Direction



To set grid parameters in the circular direction, activate the **In Circular Direction** switch of the **Direction Selection** group. Once this switch is active, the Property Bar will display the controls for location of array instances in the circular direction.



Enter the number of circular grid beams into the **N 2** field. In other words, in this field you should specify how many copies shall be created on each circle of the grid.

If this number exceeds one, control features to specify grid step in the radial direction shall become accessible.

Enter the angle between grid beams into the **Step 2** field. In other words, specify the angle step between copies along the circular direction in this field. The value of this parameter cannot exceed 360°.

Activate the switch matching the specified value of step in the **Mode 2** group (see Table 2.7.6).

Tab. 2.7.6. Mode to specify a grid step

	Value of Mode option	Method to specify a step
	Step between adjacent instances	The system views the step value as the angle between grid neighboring beams*. This option is convenient for placing a certain number of copies at the known angle between them.
	Step between boundary instances	The system views the step value as the angle between the first and the last beams of the grid*. This option is convenient for spacing a certain number of copies evenly in circle or arc.

* If the number of copies along the circular direction is equal to two, then the result of the construction shall be the same for any position of the **Mode** switch.



The same results of construction an array can be reached under various positions of the **Mode** switch. To do this, you should take into account the following correlation:

$A = (N - 1) \cdot a$, where

A is the angle step between the boundary instances, and $A < 360^\circ$

N is the number of copies

a is the angle step between neighboring instances.

If $A = 360^\circ$, then copying is done along the whole circle, and

$A = N \cdot a$



Select the direction for adding instances against the elements to be copied: **Direct** or **Reverse**. To do this, activate the applicable switch in the **Direction** group. A displayed arrow indicates direction set by the currently active switch.



2.7.4.4. Grid Parameters in Radial Direction



To set grid parameters in the radial direction, activate the **In Radial Direction** switch of the **Direction Selection** group. Once this switch is active, the Property Bar will display the controls for location of array instances in the radial direction.



Enter the number of the circular grid circles into the **N 1** field. In other words, specify in this field how many copies should be obtained on each beam of the grid.

If this number exceeds one, control features to specify grid step in the radial direction shall become accessible.

Enter the distance between circles of the grid into the **Step 1** field. In other words, specify the step between copies along the radial direction in this field.

Activate the switch matching the specified value of step in the **Mode 1** group (see Table 2.7.7).

Tab. 2.7.7. Mode to specify a grid step

	Value of Mode option	Method to specify a step
	Step between adjacent instances	Value of step is viewed by system as the distance between neighboring circles of the grid*. This option is convenient for placing a certain number of copies at a definite distance from each other.
	Step between boundary instances	The system views the step value as the distance between extreme circles of the grid*. This option is convenient for placing a certain number of copies on a line of the specific length.

* If the number of copies along the radial direction is equal to two, then the result of the construction shall be the same for any position of the **Mode** switch.



The same results of construction an array can be reached under various positions of the **Mode** switch. To do this, you should take into account the following correlation:

$$L = (N - 1) \cdot I, \text{ where}$$

L is a step between outer instances,

N is the number of copies

I is the step between adjacent instances.




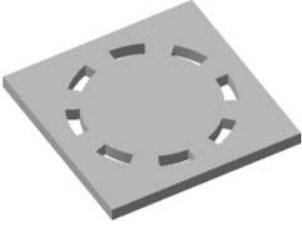

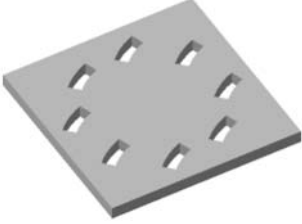
By default, instances are added against copied objects in the direction off the grid's center. The direction is **Direct**. To add instances centerwise, activate the **Reverse Direction** switch in the **Direction** group.



2.7.4.5. Array instances orientation

Instances in the array may keep the initial orientation or turn taking into account the grid angle step. To select a construction option, activate the required switch in the **Orientation** group (see Table 2.7.8).

Tab. 2.7.8. Orientation of circular array features

The value of the Orientation option	Procedure to rotate instances	Construction result
	<p>Each instance is rotated in the grid's plane around its base point so that it is positioned against its ray the same way as the base instance is positioned against its ray.</p>	
	<p>No instances rotation can be done.</p>	

2.7.4.6. Instance displacement along the array's axis

You can move instances along the **array's axis**, for example, as it is shown in Fig. 54.4.



Fig. 2.7.26. An example of an array along circular grid with instances' offset along the axis

In order to move instances along the **array's axis**, a number should be entered in the **Step along axis** field. The step value can be both positive and negative.

Positive axis direction is indicated by a straight arrow on the array's ghost.

If the step is equal to zero, all instances are in the same plane (Fig. 2.7.27, a).

If the step is not zero, the instances are moved along the array's axis (Fig. 2.7.27, b).

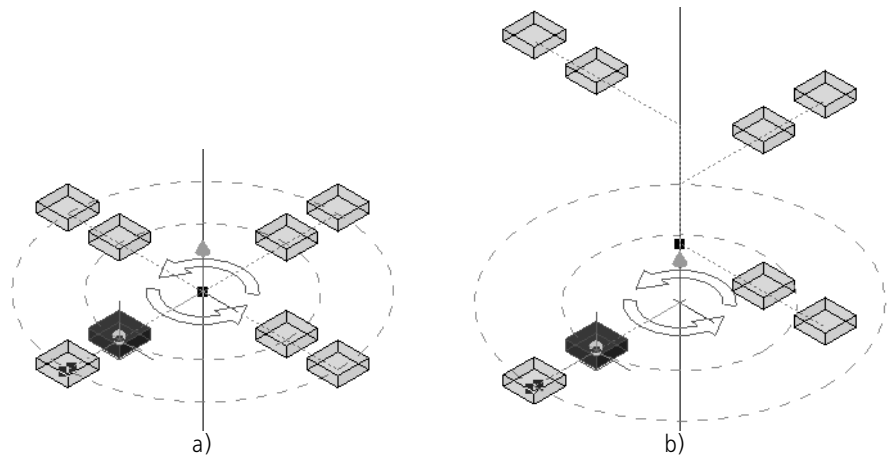


Fig. 2.7.27. Changing the array instances' positions when setting a step along the axis
a) zero step; b) step along the axis is different from zero.

The offset **T** value is calculated by the following formula:

$T = t \cdot (N - 1)$, where:

N is the number of the copy in the circular direction,

t is a step along the arrays' axis.

2.7.4.7. Changing positions of instances in an array against the base one

By default, the number of the base instance (the instance that corresponds to the object to be copied) is 1.1 (see Fig. 2.7.28, a). Array instances numbers are described in section 2.7.1.1.2 on p. 510.

You can change the array instances' position with respect to the base instance by assigning a certain number to it in the array (see Fig. 2.7.28, b).



Fig. 2.7.28. Changing instances' position in an array against the base instance
(the base instance is highlighted in white color): a) base instance with number (1.1);
b) base instance with number (1.2)

- ▼ To change the base instance's number in an array, enable the **Base instance** option on the property bar. Fields **Number 1** and **Number 2** will appear on the property bar. In these fields, enter the required numbers of the base instance in the circular and radial directions or set them


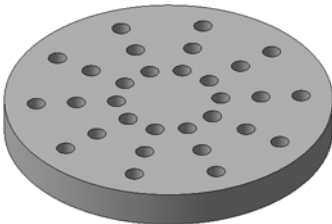

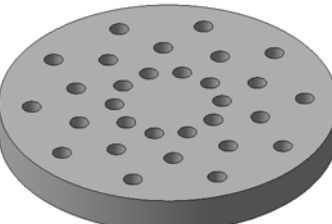

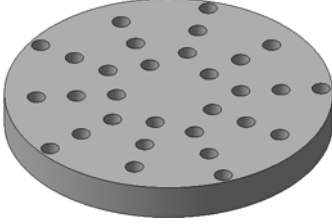
with counter. The array's ghost will be re-build – the array's instances are positioned in such a way that the number of the base instance in the array corresponds to a given number.

- ▼ To restore the base instance's default number 1.1, disable the option. The array's ghost will be rebuilt– the instances will be positioned so that the base instance number is used by default.

2.7.4.8. Positioning of copies inside the circular grid

Array's instances can be moved by half the step in the ring and radial directions. In order to specify the required variant of copies' positioning, select a line in the **Scheme list** (Table 2.7.9).

Tab. 2.7.9. Control over copies inside the circular grid

	Positioning scheme	Procedure to create an array	Construction result
	Standard scheme	Array's instances are positioned without an offset with respect to the defined grid's nodes.	
	Staggered order — displacement in the ring direction	Instances of ring rows are displaced along the nodes of the defined circular grid along the ring direction. The offset value is equal to half the distance of the arc's length between adjacent nodes of the grid. The displacement takes place in each even row in the instance building direction. The first row is the row containing the instance number (1, 1).	
	Staggered order — displacement in the radial direction	Instances of radial rows are displaced along the nodes of the defined circular grid by half the step along the ray. The displacement takes place in each even row in the instance building direction. The first row is the row containing the instance number (1, 1).	



The position of the reference instance if positioned in chess-board fashion will not be changed.



Variants of instances' positions in the staggered order make sense only in those cases when the number of instances in both direction is two or more.

2.7.4.9. Setting the position of the base instance's reference point

Base point position can be **autodefined** or specified **manually**.

The autodefine base point of the base instance is the **base point of the object to be copied** (see Section 2.1.1.4 on p. 104).

You can select the reference point setting method in the **Parameters** tab of the Property bar using the **Base point** group of switches.



By default, the base instance's reference point is the reference point of the object being copied. In this case, the **Base point** switch is active in the group **Autodefinition**.



In order to manually set the reference point's position, enable the **Manual** setting switch. The **Base point of the base instance** switch appears on the Property Bar (automatically enabled) together with the **Object** informational field. Specify the required point object in the Model Tree or window (point objects are listed in Table 2.1.7 on p. 131). The name of the selected object shall be displayed in the field beside the switch.



When copying a group of objects by circular grid, there certain peculiarities depending on whether the reference point is autodefined or set manually (see Section 2.7.4.1 on p. 543).

2.7.5. Array along the curve

You can create an array of objects by positioning them along the specified curve, for example, as is shown on the Fig. 2.7.29.

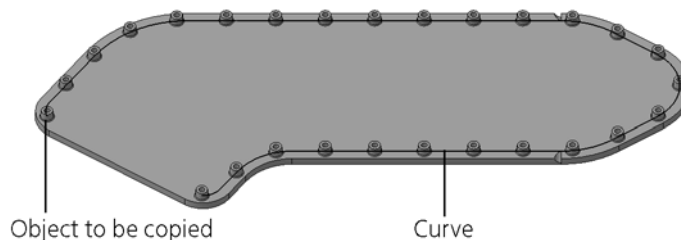


Fig. 2.7.29. Example of a Path Array

The array creation scheme is described in Section 2.7.5.1 on p. 554, array construction procedure — in Section 2.7.5.2 on p. 555.

2.7.5.1. Array construction scheme

The array along the curve is constructed in a different way depending on if the base point of the base instance is set or not.

If the base point is **set**, the array will be constructed as follows:

1. **Copying path** is found.
 - 1.1. A copy of the specified curve will be built. a curve is moved parallel to itself to make its starting point coincide with the **base point of the base instance (B)**.
 - 1.2. A segment is created to connect the base point of the base instance (B) with the **base point of the copied object (A)**.
 - 1.3. A **swept surface** is built by moving the AB segment along the curve's copy. The surface's edge opposite to the curve's copy is the copying path.
2. The array's instance is considered a separate object with a base point — **base point of the copy**. Base points of copies are positioned on the copying path in accordance with the preset copying step.
3. If additional turn is enabled, each copy is rotated around its base point (see. Section 2.7.5.6 on p. 559).

Diagram of the array construction in the above-described way is detailed in Fig. 2.7.30, a.

If the base point is **not set**, the array will be constructed as follows:

1. **Copying path** is found. To do this, a copy of the curve is created: a curve is moved parallel to itself to make its starting point coincide with the **base point of the copied object (A)**. The resulting copy of the curve will be the copying path.
2. Base points of copies (see above) are positioned on the copying path in accordance with the preset copying step.
3. If additional turn is enabled, each copy is rotated around its base point (see. Section 2.7.5.6 on p. 559).

Diagram for constructing the array in the above-described way is detailed in Fig. 2.7.30, b.

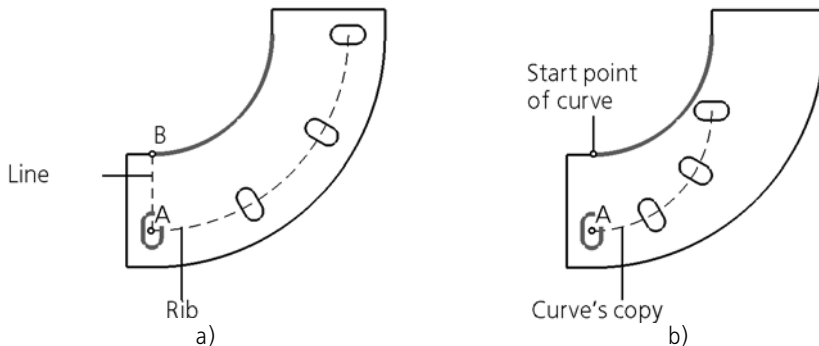


Fig. 2.7.30. Diagram for copying the extrusion feature along the whole guiding line while maintaining the orientation (path and the copied object are highlighted):
 a) point **B** is chosen as the base point of the base instance,
 b) position of the base instance's base point is not set

2.7.5.2. Procedure of the array creation along the curve

To create a path array, perform the following steps.



1. Call the **Array along curve** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).

2. Specify the **copied object**: operations or bodies or surfaces or curves, points, auxiliary planes and axes, or components.

For details about copied object selection, see Section 2.7.1.3.1 on p. 512.

The names of the selected objects will appear in the **List of Objects** toolbar.

The ghost of the **reference point of the object being copied** appears on the screen as three mutually perpendicular vectors (see Section 2.1.1.4 on p. 104).

Parameters, Deleted, Properties tabs will appear on the property bar.



You can also select objects for copying prior to calling the array creation command.

3. Go to the **Parameters** tab.
 - 3.1. Specify a **curve** to create an array.

The following may be used as this curve:

- ▼ a 3D curve,
- ▼ an edge,
- ▼ a sketch line,
- ▼ a chain of objects listed above in any combination.



Enable the **Curve** switch and specify the required object or objects in the Design Tree or in the model window. The name of the selected object or the number of selected objects are shown in the **Curve** field. To cancel object selection, re-select it.

The array ghost will be displayed. The starting point of the specified curve is used as the **base point of the base instance** (the extreme point of the curve nearest to the copied object). It depends on mutual position of the copied object and the starting point of the curve (see Section 2.7.5.7 on p. 559).

- 3.2. If necessary, set the **starting point** of the curve manually, if the copying path is closed (see Section 2.7.5.3 on p. 556).
- 3.3. Enter the number of copies into the **Number** field.
- 3.4. Specify the **copying step** (see Section 2.7.5.4 on p. 556).
- 3.5. If necessary, use the switches of the **Direction** group to change the **direction of copying** suggested by the system (see Section 2.7.5.5 on p. 558).
- 3.6. Select **Orientation of instances** (see Section 2.7.5.6 on p. 559).



All the changes will be reflected in the array ghost.

4. If required, you can also:
 - ▼ Changing the position of the **base instance's reference point** (see Section 2.7.5.7 on p. 559).

- ▼ Disabling using the **projection of the reference point of the object being copied — available, if rotation bodies/surfaces are copied** (see Section 2.7.1.3.3 on p. 514).
- ▼ Creating a **geometric array** — available, if operations are copied (see Section 2.7.1.3.5 on p. 515).
- ▼ Deleting and restoring array instances. Enable the **Instances** tab of the Properties Bar. See Section 2.7.1.3.6 on p. 516 for information on the deletion and restoring array's instances.
- ▼ Managing form and dimensions of instances by changing their variables' values (see Section 2.7.2 on p. 519).
- ▼ Changing of array name and properties of surface of instances. Enable the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.

2.7.5.3. Curve start point

The starting point of an **open** path is its end point closest to the object being copied.

The starting point of a **closed** path is its first vertex (for a chain of objects— the vertex nearest to the copied object).

If necessary, you can **manually** define the starting point of a closed path. You can specify any point on the path as the starting point.



Activate the **Start point** switch on **Parameters** tab of Properties toolbar and select the required point in the model window. The name of the selected object is displayed in a field beside the switch. To cancel object selection, re-select it.

Dependence of array instances disposition from the starting point selection is shown in Fig. 2.7.31.





Fig. 2.7.31. Relation between disposition of copies and location of the start point of the curve.
(copying path and the object to be copied are highlighted, starting point is indicated by a circle)

2.7.5.4. Copying step



Copies can be placed evenly along the whole guiding line or at a specified distance from each other. To select a construction option, activate the applicable switch in the **Method** group (see Table 2.7.10).

Tab. 2.7.10. Methods to set array step

Value of option Method	Concept of determining the copying step
 By Step	<p>You can set a step value and select a mode of step determination* :</p> <ul style="list-style-type: none"> ▼ between neighboring copies or ▼ between boundary copies. <p>Enter into Step field the value of a step between copies which is measured along the copy path.</p> <p>Activate the switch matching the specified value of step in the Mode group (see Table 2.7.11).</p>
 Along the Entire Guiding Line	<p>Full step is determined automatically as the length of the copying path. The first and the final array instances lie at the start and the end points of the guiding curve. Array instances are spaced evenly along the path. Field Step and group Mode are not present on the property bar.</p>

* If the value of full step is set as equal to the length of path, then the result of construction shall be the same as for the **Along Whole Guiding Line constructing** method.

Tab. 2.7.11. Mode of setting step along the path

Value of Mode option	Method to define a step
 Step between adjacent instances	<p>The system views the step value as spacing between corresponding insertion points of neighboring copies in the direction of copying* . This option is convenient for placing a certain number of copies at a definite distance from each other.</p>
 Step between boundary instances	<p>The system views the step value as spacing between corresponding insertion points of the first and the second copies in the direction of copying* . This option is convenient for placing a certain number of copies on a line of the specific length.</p>

* If the number of copies is equal to two, then the result of construction is the same for any position of the **Mode** switch.



The same results can be reached under various positions of the **Mode** switch. To do this, you should take into account the following correlation:

$$L = (N - 1) \cdot I, \text{ where}$$

L is the step between the extreme instances measured along the path

N is the number of copies

I is the step between neighboring instances measured along the path.

2.7.5.5. Direction of copying along the path



While creating an array along curve, you can direct the copying using switches of the **Direction** group.

A change in the direction has a different effect on arrays with closed and open path.

If the path is closed, then the **Forward Direction** of copying means placing array instances along the path on the same side of the copied object, while the **Reverse Direction** means placing them on the other side (Fig. 2.7.32). The current copying direction is shown by the arrow on the array ghost.

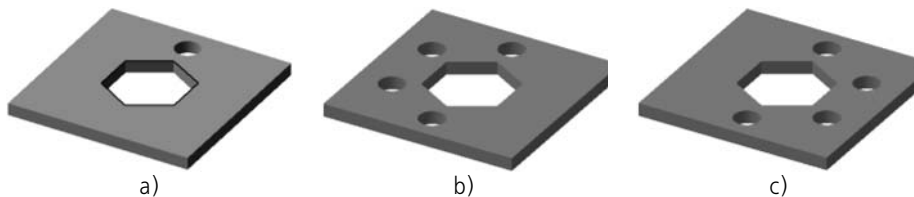


Fig. 2.7.32. Change of direction for the array with closed path:
a) copied object and path (marked with a thicker line),
b) array in forward direction, c) array in reverse direction



If the copying path is closed, and instances are positioned along the whole guiding line (see Table 2.7.7 on p. 549), the operation's result does not depend on the copying direction.

If the path is open, then the **Forward Direction** of copying means the start of the path is defined by default and the starting point is the path's end nearest to the copied object. When selecting the **Reverse Direction**, the other end will be considered as the path start (Fig. 2.7.33).

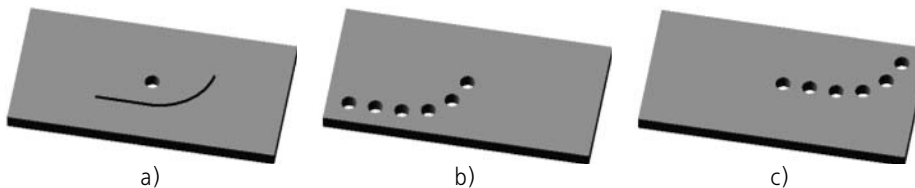






Fig. 2.7.33. Change of direction for the array with an open path:
a) the copied object and path,
b) array in forward direction, c) array in reverse direction

2.7.5.6. Orientation of Array Instances

Copies in the array can maintain the initial orientation or rotate taking into account the curvature of the path. To select a construction option, activate the required switch in the **Orientation** group (see Table 2.7.12).

Tab. 2.7.12. Copies Orientation along the Path

	The value of the Orientation option	Procedure to rotate copies	Construction result
	Save initial orientation	No copies rotation can be done.	
	Set Radially to Normal	Each copy is rotated so that it gets oriented with respect to the tangent path vector in its base point, the same way as the copied object is oriented with respect to the tangent path vector in its base point.	

2.7.5.7. Setting the position of the base instance's reference point

Base point position can be **autodefined** or specified **manually**.

The autodefine base point of the base instance is the **base point of the object to be copied** (see Section 2.1.1.4 on p. 104).

The array building result depends on the method of defining the reference point (see Section 2.7.5.1 on p. 554).

You can select the reference point setting method in the **Parameters** tab of the Property bar using the **Base point** group of switches.



If the reference point of the object being copied coincides with the curve's starting point and is positioned on the tangent to the curve in this point, the **Autodefine** switch is enabled by default.



In other cases, the **Manual setting** switch is enabled. And the starting point of the specified curve is suggested as the base point of the base instance (the extreme point of the curve nearest to the copied object). The reference point's name will be displayed in the **Object** field.



If it is necessary to change the base point, enable the **Base point of the base instance** switch in the **Base point** group and specify the required point object in the Design Tree or model window (point objects are listed in Table 2.1.7 on p. 131).

If necessary, you can change the reference point setting method. To do this, enable the respective switch. **Autodefine** or **Manual setting**.



To get a predictable result of copying:

- ▼ when using the manual definition of the base point, it is recommended to select the path start as the base point.
- ▼ when using auto determination of the base point, it is recommended to create the path in such a way that it would definitely start in the base point of the object being copied.

2.7.6. Pattern by Points

You can create an array of objects, whose instances' positions are set by point objects (Fig. 2.7.34). Point objects are listed in Table 2.1.7 on p. 131.

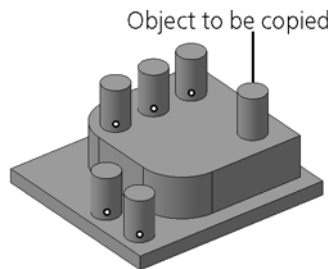


Fig. 2.7.34. Example of an array by points

2.7.6.1. Creating an array by points

To create an array by points, complete the following steps.



1. Call the **Array by Points** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).

2. Specify the **copied object**: operations or bodies or surfaces or curves, points, auxiliary planes and axes, or components.

For details about copied object selection, see Section 2.7.1.3.1 on p. 512.

The names of the selected objects shall appear in the **List of Objects** toolbar.

The ghost of the **reference point of the object being copied** appears on the screen as three mutually perpendicular vectors (see Section 2.1.1.4 on p. 104).

Parameters, Deleted, Properties tabs will appear on the property bar.



You can also select objects for copying prior to calling the array creation command.

3. Go to the **Parameters** tab.



- 3.1. Specify **positions** of the array's instances. To do this, activate the **Points** switch and specify the required point objects in the Design Tree or in the model window (point objects are listed in Table 2.1.7 on p. 131). Specified point objects are color highlighted.

The name of the selected object or the number of selected objects are shown in the **Points** field. To cancel object selection, re-select it.

The array ghost will be displayed: the base point of the base instance is the base point of the copied object; base points of other instances of the array will coincide with the indicated points.

If array instance positions are set by the group points or sketch points, you can select the required points in the Design Tree or in the model window. To select all points of group or sketch, specify this group or sketch in the Design Tree.

If the sketch specified in the Design Tree is used to specify instances' positions, you can control the plane where these instances are positioned (see Section 2.7.6.2 on p. 561).

3.2. Select **Orientation of instances method** (see Section 2.7.6.3 on p. 562).

All the changes will be reflected in the array ghost.

4. If required, you can also:

- ▼ Changing the position of the **base instance's reference point** (see Section 2.7.4.9 on p. 553).
- ▼ Disabling using the **projection of the reference point of the object being copied; it is available, if rotation bodies/surfaces are copied** (see Section 2.7.1.3.3 on p. 514).
- ▼ Creating a **geometric array** — available, if operations are copied (see Section 2.7.1.3.5 on p. 515).
- ▼ Deleting and restoring array instances. Enable the **Instances** tab of the Properties Bar. See Section 2.7.1.3.6 on p. 516 for information on the deletion and restoring array's instances.
- ▼ Managing form and dimensions of instances by changing their variables' values (see Section 2.7.2 on p. 519).
- ▼ Changing of array name and properties of surface of instances. Enable the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.

Number of copies of the source object is equal to the number of points defining array instances positions (n).

If one of copies coincides with the source object, this copy will not be created. Number of one object copies is equal to $n-1$.

2.7.6.2. Control of the plane of the array instances' position

When setting positions of the array's instances, you can use the sketch with points. In this case, all points of the sketch will be used for the array creation.



To set sketches' positions using all points of the sketch, enable the **Points** switch on the **Parameters** tab and specify the required sketch in the Design Tree.

The **Project** option will appear on the property bar that controls the plane of the instances' positions set by the points of the specified sketch.

If the option is enabled, the array will be built as follows (Fig. 2.7.35,a).

1. An additional plane is created parallel to the sketch plane and passing through the base *C* point of the object of copying.
2. Projections of sketch points to the additional plane are created.
3. Instances are positioned so that their base points coincide the projected points of the sketch on the additional plane.

If the option is disabled, the instances of the array are positioned so that their base points coincide the points in the sketch (Fig. 2.7.35,b).

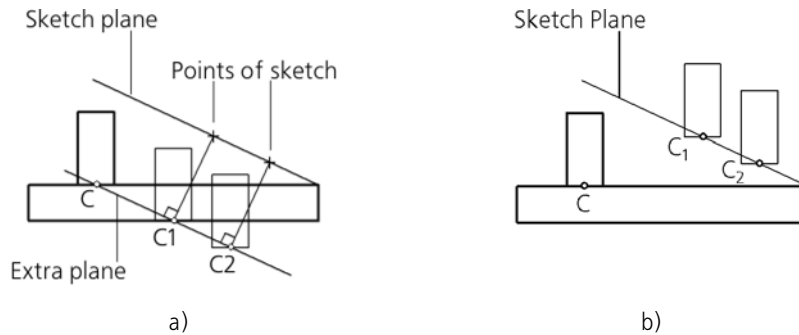


Fig. 2.7.35. Setting positions of array instances:
a) if the **Project** option is enabled; b) if the **Project** option is disabled

2.7.6.3. Orientation of Array Instances

Array's instances can maintain their initial orientation or be rotated.

To select an instance construction option, activate the required switch in the **Orientation** group on the Property Bar:



- ▼ **Save initial orientation** (see Section 2.7.6.3.1 on p. 562),



- ▼ **Orient by object related to point** (see Section 2.7.6.3.2 on p. 563),



- ▼ **Orient by specified object** (see Section 2.7.6.3.3 on p. 564).

2.7.6.3.1. Construction of instances with initial orientation

You can create array instances with their initial orientation. Each instance's orientation in its base point will be the same as orientation of the copied object in its base point (Fig. 2.7.36).

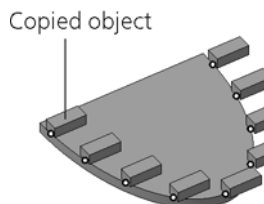


Fig. 2.7.36. Positioning array instances with their initial orientation



In order to position instances with their initial orientation, enable the **Keep initial orientation** switch in the **Orientation** group on the Property Bar. After enabling the switch, all instances will be oriented in space in the same way as the copied object.

2.7.6.3.2. Construction of instances with orientation by object linked to points

Positions of array instances may be defined by points on the curve or on the surface. Such points include:

- ▼ points built by the **Group of points on curve** command or **Group of points on surface** (see Sections 2.5.3.2 on p. 350 and 2.5.3.3 on p. 353);
- ▼ points built by the **Point** command using the **On curve** or **On surface** method (see Sections 2.5.2.5 on p. 338 and 2.5.2.6 on p. 339);



If positions of instances are defined by the above-mentioned points, you can orient instances with respect to the object(s) on which points are positioned. For purpose, use the **Orient by object linked to point** switch of the **Orientation** group on the Property Bar.

Once the switch is enabled, instances will be oriented in the following way:

▼ Points on curve

Each instance is rotated around its reference point so that it gets oriented with respect to the tangent curve vector in this point, the same way as the copied object is oriented with respect to the tangent curve vector in this curve's point which is closest to the reference point of the object being copied (Fig. 2.7.37, a).

▼ Points on surface

Each instance is rotated around its reference point so that it gets oriented with respect to the surface's normal in this point, the same way as the copied object is oriented with respect to the surface's normal in this curve's point which is closest to the reference point of the object being copied (Fig. 2.7.37, b).



Fig. 2.7.37. Example of array instances positioning:
a) with orientation by curve; b) with orientation by surface



The set of points that define positions of instances may include points that are positioned on a curve or surface. In this case, once the **Orient by object linked to point** switch is enabled, the instances with positions defined by these points will be oriented in space in the same way as the copied object.



The **Orient by object linked to point switch** is not available, if

- ▼ all points that define positions of instances are not positioned on a surface or curve,
- ▼ all points that define positions of instances are positioned on a flat surface or linear curve,
- ▼ the base point of the object being copied is not projected on the surface or curve to which the points belong.

2.7.6.3.3. Construction of instances with orientation by specified object

Array's instances can be oriented by a normal to the specified object. The following may be used as the object:

- ▼ an edge, 3D curve, or sketch line (except for straight lines);
- ▼ face (except for flat ones).

Example of positioning array instances with orientation by a spheric face is shown in Fig. 2.7.38.

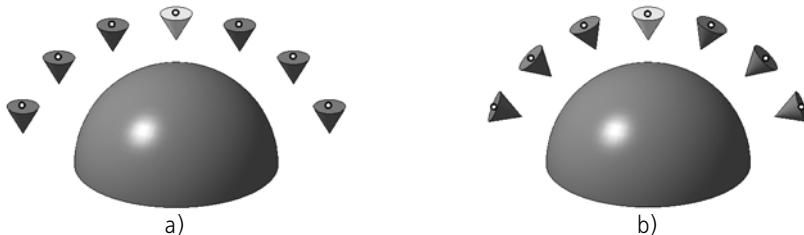


Fig. 2.7.38. Example of array instances positioning
(the base instance is highlighted):

a) with the initial orientation retained; b) with orientation by spherical face



In order to change orientation of instances in space with respect to an object, enable the **Orient by specified object** switch. The **Object** field will appear next to the switch on the Property Bar.

Specify the required object. The specified object is highlighted by a certain color, and its name is displayed in the **Object** field.

Orientation of the instance is defined as follows.

1. The nearest projection of the point, that defines the instance's position, to the specified object will be built (Point *A*).
2. The nearest projection of the base point of the base instance to the specified object will be built (Point *B*).
3. The instance is rotated around its base point so that it is oriented with respect to the normal to the specified object in the *A* point in the same way as the copied object is oriented with respect to the normal to the specified object in point *B*.



If no projections on the specified object are built for points that define the instances position, the instances related to these points will be oriented in space in the same way as the copied object.

2.7.7. Array by Table

You can create an object array, which instances positions are set by points. Positions of points, in their turn, can be set using two methods:

- ▼ by reading coordinates saved in the data file (see Appendix VI for more information about the data file),
- ▼ manual entry of coordinates in the table of modifiable variables which is saved in the model.

Values read from the data file or specified in the table of modifiable variables can be interpreted as rectangular, cylindrical, or spherical coordinates of points (more information about types of a point's coordinates can be found in Section 2.5.3.4.1 on p. 358).

If instances' positions are specified using the data file, link between the created array and the data file will be kept.

Another way of using the **Array by points** command is to get an array of objects which differ in parameters only (i.e. only values of their variables differ). For example, you can build copies of the extrusion operation which have differences only in the following parameters:

- ▼ values of dimensions which define the position of a contour in a sketch counting from the zero point of the coordinates of this sketch (dimensions in the sketch must be set before the array is created),
- ▼ values of a variable which defines the extrusion depth.

The resulting extrusion operations will apply to the same section but will differ in length and position.

Table array relates to objects that use coordinate system (for object coordinate system refer to Section 2.8.3.4.1 on p. 598).

2.7.7.1. Creation of an array using table

You can create an array from a table in two ways:

1. An array of objects of a specific type, e.g. operations. In this case you will be able to set the positions and orientation of the instances. You can change the values of the variables for instances if required. And vice versa, you can set only values of variables without setting their positions.
2. An array of an arbitrary objects. In this case you will not be able to set the positions and orientation of the instances To avoid duplicating of the instances, change the values of their variables.



The limitation of the second option is that you cannot copy bodies, components and previously created arrays.

Option 1 Array of objects of selected type



1. Call the **Array by table** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).

2. Specify the **copied object**: operations or bodies or surfaces or curves, points, auxiliary planes and axes, or components.

For details about copied object selection, see Section 2.7.1.3.1 on p. 512.

The names of the selected objects will appear in the **List of Objects** toolbar.

The ghost of the **reference point of the object being copied** appears on the screen as three mutually perpendicular vectors (see Section 2.1.1.4 on p. 104).

Parameters, Deleted, Properties tabs will appear on the property bar.



You can also select objects for copying prior to calling the array creation command.

3. Go to the **Parameters** tab.
 - 3.1. Set **positions** of instances (see Section 2.7.7.2 on p. 568).
 - 3.2. Select the **coordinate type** for interpretation of data (from the table of modifiable variables or from the data file; more information about types of coordinates of a point can be found in Section 2.5.3.4.1 on p. 358). To do this, activate the relevant switch in the **Coordinate Type** group on the Properties bar:



- ▼ **Rectangular X,Y,Z,**



- ▼ **Cylindrical R,A,Z,**



- ▼ **Spherical R,A,B**

Points for setting the positions of instances will be built in the coordinate system which corresponds to the selected type.



Note that values of coordinates will be kept in case of changing their type. These values will be assigned to coordinates of the selected type.

If the type of coordinates is changed when working with the table of modifiable variables (the **Table of modifiable variables** window is open), you can:

- ▼ *save values of coordinates* – assign current values of coordinates to coordinates of the selected type,
- ▼ *recalculate coordinate values* – current values of coordinates will be changed in accordance with the selected type of coordinates.

A dialog will appear allowing you to choose the desired action. Click the **Yes** button to save the coordinate values or the **No** button to recalculate coordinate values.

- 3.3. Select **Orientation of instances method** for the array (see Section 2.7.7.4 on p. 574). All the changes will be reflected in the array ghost.
4. If required, you can also:
 - ▼ Changing the position of the **base instance's reference point** (see Section 2.7.4.9 on p. 553).
 - ▼ Disabling using the **projection of the reference point of the object being copied; it is available, if rotation bodies/surfaces are copied** (see Section 2.7.1.3.3 on p. 514).

- ▼ Creating a **geometric array**; it is available, if operations are copied (see Section 2.7.1.3.5 on p. 515).
- ▼ Deleting and restoring array instances. Enable the **Instances** tab of the Properties Bar. See Section 2.7.1.3.6 on p. 516 for information on the deletion and restoring array's instances.
- ▼ Changing of array name and properties of surface of instances. Enable the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.

Option 2 An array of an arbitrary objects



1. Call the **Array by table** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).



2. Specify the **objects to copy**.

2.1. In the **Type** group, enable the **Arbitrary** switch. **Parameters** and **Properties** tabs will appear on the Property bar.

- 2.1. Specify the required object in the Design Tree or Model Window.

If you choose objects using the Tree, then the objects between the specified ones will also be included when the **Select the intermediate items** option is enabled¹ (when the Tree displays the sequence of steps for building a model).

The names of the selected objects will appear in the **List of Objects** toolbar.



3. Set the variables related to the position, form and dimensions of the instances. To do this, click the **Table of modifiable variables** button on the Special Control Panel. In the table that appears add a line for each instance to be built, choose the modifiable variables and set their values (see Section 2.7.2 on p. 519 for details).

4. If necessary, change the name and properties of the created surface. Enable the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.



When using method 2, no ghost of the array is shown.

You can remove or restore the array instances as required. To remove a previously created array, choose the instance you want to delete and then press *<Delete>*. To restore the removed instances of an array when editing, use the **Instances** tab on the Properties bar. See Section 2.7.1.3.6 on p. 516 for information on the deletion and restoring array's instances.

1. The objects which cannot be copied are not included into the selection.

2.7.7.2. Setting positions of instances

Positions of points, that define positions of instances, can be set using two methods:

- ▼ by reading coordinates saved in the data file (see Appendix VI for more information about the data file),
- ▼ by manual entry of coordinates in the table of modifiable variables, which is saved in the model (see Section 2.7.2.2 on p. 521 for more information about the table of modifiable variables).

The **Source of data** group of switches on the **Parameters** tab makes it possible to choose the method of specifying instances' positions:



- ▼ **Source file** – reading data from file (see Section 2.7.7.2.1 on p. 568),



- ▼ **Manual entry** – manual entry of data in the table of modifiable variables (see Section 2.7.7.2.2 on p. 569).

Peculiarities of using the table of modifiable variables are described in Section 2.7.7.3 on p. 572.

2.7.7.2.1. Reading data from file

This method makes it possible to specify instances' positions by points the coordinates of which are obtained from a previously created file (see Appendix VI for more information about the data file).



To set instances' positions, complete the following steps.

1. Go to the **Parameters** tab of the Property Bar and click the **Source file** button in the **Data source** group.
2. The standard Windows File Open dialog shall appear on the screen. Specify a file name and click the **Open** button. The name of the file will appear in the **Source File** field.

The array ghost will be displayed: the base point of the base instance is the base point of the copied object; base points of other instances of the array will coincide with the points whose coordinates are obtained from the data file. These points will be highlighted with a color.

More information about the base point of the copied object can be found in Section 2.1.1.4 on p. 104; see Section 2.7.1.1.1 on p. 510 for more information about the base point of the array's instance.

You can view data read from file in the **Table of modifiable variables** window. For this, do the following.



1. Click the **Table of modifiable variables** button on the Special Control Panel. The **table of variables** will be displayed on the screen.



2. If no coordinates are listed in the table, enable them by clicking **Set coordinates of the base point** on the window toolbar.

Columns with coordinates of instances' base points appear in the table of modifiable variables (Fig. 2.7.39). Coordinate values of the copied object's base point are displayed in the second row. Each row of the table, starting from the third one contains the instance's number and coordinates of the point defining its position.

Note that columns with coordinates are displayed in gray color. It means that values of coordinates are not available for editing.

Instance	X	Y	Z
(1)	-15	30	0
(2)	-21	120	94
(3)	-94	57	53
(4)	5	102	73
(5)	16	96	23

Fig. 2.7.39. Example of the table of modifiable variables with coordinates obtained from file

The link between the created array and the data file is maintained. This link is, in general, similar to the link between a group of points and the source file (see Section 2.5.3.4.2 on p. 359). The difference is in link management: it is impossible not to create it during array construction (no **Explode** option on the Property Bar).

2.7.7.2.2. Manual entry of data in the table

This method makes it possible to set instances' positions by manual entry of coordinates of instances' base points in the table of modifiable variables (see Section 2.7.2.2 on p. 521 for information about the table of modifiable variables).

To set instances' positions, complete the following steps.



1. Go to the **Parameters** tab of the Property Bar and click the **Manual entry** button in the **Data source** group. The **Table of modifiable variables** command will be run automatically (its button can be found on the Property Bar).



The **Table of modifiable variables** appears on the screen with the table having the following structure (initial layout of the table; Fig. 2.7.40):

- ▼ The first row contains the names of coordinates (headers of columns).
- ▼ Coordinate values of the copied object's base point are displayed in the second row.
- ▼ The remaining rows of the table contain numbers of instances and empty cells (by default, three rows are created in the table).



An empty cell of the table of modifiable variables means that the coordinate in the cell has the same value as the value of the respective coordinate of the base instance's point.

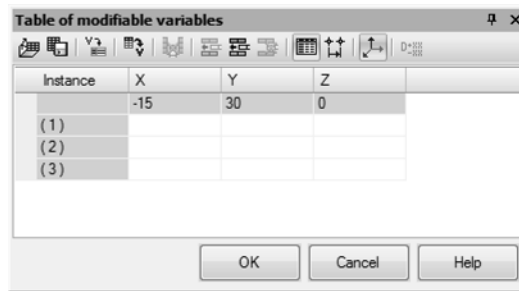


Fig. 2.7.40. Example of the table of modifiable variables for entry of coordinates (initial layout of the table)



Note that coordinates are displayed in the table if the **Set coordinates of the base point** button is enabled in the toolbar of the **Table of modifiable variables** window.

2. Specify the required number of instances (rows in the table) and enter new coordinates of the instances' base points in the table. New coordinates can be entered using two methods:
 - ▼ by entering coordinates in the table's cells (Method 1),
 - ▼ by specifying an increment value for coordinates (Method 2).
 Methods of filling the table are described in detail in this section below.
3. Click **OK** to close the **Table of modifiable variables** window.

The table with coordinates of instances' base points is added to the current file.

The array ghost will be displayed: the base point of the base instance is the base point of the copied object; base points of other instances of the array will coincide with the points whose coordinates are set in the table of modifiable variables.



If a value outside the range is entered in the table's cell, the table will not be closed after clicking **OK**. A message will appear informing that the values is outside the range, and the cell's borders will be highlighted. The range of acceptable values depends on the parameter to which it corresponds. For example, this message may appear if a negative value is entered in the cell of the *B* coordinate of the spherical system of coordinates, since the range of acceptable values of this coordinate is $0^\circ \dots 180^\circ$.

See Section 2.7.2.2.3 on p. 524 for details about working with the table of modifiable variables.



Editing coordinates in the table of modifiable variables is the same as their entry. In order to begin editing the table, click the **Table of modifiable variables** button on the Special Control Panel.

Methods of filling the table of modifiable variables by coordinates of points that define instances' positions

Method 1

Entry of coordinates of points that define instances' positions in the table of modifiable variables.



1. In order to use this method, click the **Values of Variables** button in the **Table of modifiable variables** window. The table of modifiable variables will appear in the window in its initial layout (Fig. 2.7.40).
2. Specify the required number of instances. For this purpose, change the number of rows in the table using the following buttons:



▼ **Add row above,**



▼ **Add row below,**



▼ **Delete rows.**



Note that cells of the new row added to the table are empty. An empty cell means that the coordinate in the cell has the same value as the value of the respective coordinate of the base instance's point.

3. Enter new coordinates of instances' base points in the table. For this purpose, activate the required cell by double-clicking it, enter the new coordinate's value and click <Enter>.

Method 2

Setting increment value for coordinates of instances' base points.



1. In order to use this method, click the **Increment value** button in the **Table of modifiable variables** window.

A table with the following structure will appear in the window (Fig. 2.7.41):

- ▼ The first row, starting from the third cell, contains the names of coordinates (headers of columns).
- ▼ The second column is used for setting the number of instances.
- ▼ The second row, starting from the third one, contains coordinates of the base point of the copied object.
- ▼ The third row is used for setting the increment value for coordinates.

	Of instances	X	Y	Z
Value	3	-15	30	0
Step				

Fig. 2.7.41. Example of the table of modifiable variables for setting the increment value of coordinates

2. Specify the required number of instances.
 - 2.1. Activate the required cell by double-clicking it in the second row and enter the required number of array instances in it.

- 2.2. Press <Enter>.
3. Specify the increment values for coordinates.
 - 3.1. In the third row of the table, activate the required cell by double-clicking it and enter the increment value for coordinates it (the increment value can both positive and negative).
 - 3.2. Press <Enter>.



If necessary, you can view the new coordinates of points. For this purpose, click the **Values of Variables** button in the **Table of modifiable variables** window. The table with coordinates will be displayed in the window. The column of the coordinate, for which increment value was assigned, will contain its new values.

2.7.7.3. Peculiarities of using the table of modifiable variables

Specifying positions of instances

One of the methods of setting positions of array's instances by table is reading coordinates of points from the data file. The table of modifiable values makes it possible to view coordinates obtained from the file.



In order to enable displaying coordinates in the table, click the **Set coordinates of the base point** button in the toolbar of the **Table of modifiable variables** window. Columns with coordinates will appear in the table (Fig. 2.7.42). Note that columns with coordinates are displayed in gray color. It means that values of coordinates are not available for editing.

Instance	X	Y	Z
	-15	30	0
(1)	379	54	-101
(2)	50	240	90
(3)	50	500	63
(4)	37	5	-10

Fig. 2.7.42. Example of the table of modifiable variables with coordinates obtained from file

Another method of specifying instances' positions is manual entry of coordinates of the instances' base points in the table of modifiable variables. Table cells for entering values of coordinates have white color (Fig. 2.7.43). It means that values in these cells can be modified. The exception is the row with coordinates of the base point of the copied object (the row is gray); these coordinates cannot be edited.

Instance	X	Y	Z
(1)	-15	30	0
(2)	-15	30	20
(3)	-25	45	25
(4)	-35	60	40
(4)	-45	75	65

Fig. 2.7.43. Example of the table of modifiable variables with coordinates set manually

Setting a new value of an instance's variable



New values of variables for array instances by table can be obtained from the data file (see Appendix VI). For this purpose, the **Read from file** button is used. The table of modified variables is filled with data from file as follows: values are entered only in the cells that are displayed in the table at the moment the data is read and are available for editing (they are white).

Let's take the table of modifiable values illustrated in Fig. 2.7.44 as an example. The table contains columns with coordinates of instances' base points and columns with values of variables for instances.

In Fig. 2.7.44, a) cells with coordinates are not available for editing, cells with values of variables are available. Values from the data file will be entered only in those cells that are available for editing (in variables' columns).

In Fig. 2.7.44, b) cells with coordinates' values and cells with variables' values are both available for editing. Values from the data file will replace the contents of all cells that are available for editing.



To keep the values of coordinates' base points unchanged, disable displaying them in the table. You can do this using the **Specify base point coordinates** button. Then, values from the data file will be entered only in the remaining cells of the table (in variables' columns).

Instance	X	Y	Z	v12	v134
(1)	379	54	-101	20	25
(2)	50	240	90		
(3)	50	500	63		
(4)	37	5	-10		

a)

Fig. 2.7.44. Example of the table of modified variables:
a) coordinates of instances' base points are not available for editing;
b) coordinates of instances' base points are available for editing

Instance	X	Y	Z	v12	v134
	-15	30	0	20	25
(1)	379	54	-101		
(2)	50	240	90		
(3)	50	500	63		
(4)	37	5	-10		

b)

Fig. 2.7.44. Example of the table of modified variables:
 a) coordinates of instances' base points are not available for editing;
 b) coordinates of instances' base points are available for editing

If the file has more lines than the table when reading the data file, then:

- ▼ For the table in which cells with coordinates values are available for editing (Fig. 2.7.44, b) a prompt to add more lines will be displayed.
Click **Yes** to add more lines. To cancel adding lines, click **No**.
- ▼ For the table in which cells with coordinates values are not available for editing (Fig. 2.7.44, a) no prompt to add more lines is displayed. Data from file are entered only in the lines available in the table.

Saving a table to data file

Only those values are saved to the data file that are positioned in cells that are available for editing. Values are saved irrespective of whether displaying of cells containing these values is enabled or disabled in the table.

We will describe how to write the table shown in Fig. 2.7.44 into a file.

In Fig. 2.7.44, a) you can see the table with cells containing coordinates that are not available for editing, while cells with variables' values are available for editing. Only variables' values will be saved to the data file.

In Fig. 2.7.44, b) you can see the table with cells containing coordinates' values and cells with variables' values that are both available for editing. Both coordinate values and variables' values will be saved to the data file. If displaying of columns with coordinates of points is disabled for a certain table, then, irrespective of this, values of points' coordinates will be saved in the data file.

2.7.7.4. Orientation of Array Instances

Array's instances can maintain their initial orientation or be rotated.

To select an instance construction option, activate the required switch in the **Orientation** group on the Property Bar:



- ▼ **Save initial orientation** (see Section 2.7.6.3.1 on p. 562),



- ▼ **Orient by radial direction** (see Section 2.7.7.4.1 on p. 575),



- ▼ **Orient by object** (see Section 2.7.6.3.3 on p. 564).

2.7.7.4.1. Construction of instances with turning in radial direction



If instances' positions are specified in cylindrical or spherical coordinates, you can orient instances in the radial direction. For this purpose, use the **Turn in radial direction** switch of the **Orientation** group on the Property Bar.

Once the switch is enabled, instances will be oriented in the following way:

- ▼ **Positions of instances are set in cylindrical coordinates**

Each instance is rotated around its base point so that it gets oriented with respect to a perpendicular from this point to Z axis, the same way as the base instance is oriented with respect to a perpendicular from its base point to Z axis (see Fig. 2.7.45, a).

- ▼ **Positions of instances are set in spherical coordinates**

Each instance is rotated around its base point so that it gets oriented with respect to a ray, going from the sphere's center to its base point the same way as the base instance is oriented with respect to its ray going from the sphere's center to its base point (see Fig. 2.7.45, b).



Fig. 2.7.45. Example of orienting array's instances in radial direction:
 a) positions are set in cylindrical coordinates;
 b) positions are set in spherical coordinates.

2.7.8. Mirror array

You can create copy of objects symmetrical to the them relative to the specified planar object.



Creation of mirror copy of assembly components is impossible.

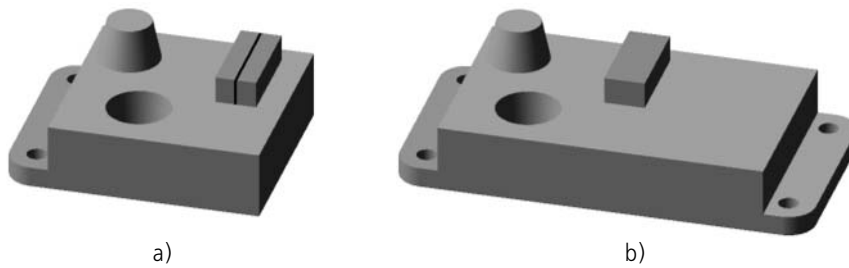


Fig. 2.7.46. Mirror copying of objects:
 a) initial part (plane of symmetry for copying is shown nominally),
 b) the result of copying the base and support
 relative to midplane of rectangular projection

- ▼ The result of mirror-copying of an operation, plane, curve, point, auxiliary plane or surface is a new object being a mirror copy of the original object and of the same type as the original object.
- ▼ The result of mirror-copying of a body is a body possessing a symmetry plane, or new body that is a mirror copy of the existing one.

Mirror array for various object types is created differently. Therefore, control elements for creating an object's mirror copy depend on the type of the object: one set of control elements for a body or surface and another set is designed for operations, curves, or points.

The procedure of construction of a mirror array for operations, curves, and points is described in Section 2.7.8.1 on p. 576; creation of a mirror copy of a body or of a surface — see Section 2.7.8.2 on p. 577.

Note that each call of this command allows you to copy one of the following:

- ▼ either one or more operations,
- ▼ either one or several objects of the following types: a curve, point, auxiliary plane or axis,
- ▼ either one body,
- ▼ either one surface.

2.7.8.1. Creation of a mirror array of operations, curves, points, auxiliary planes, and axes

To create a mirror array of operations, curves, points, auxiliary planes or axes, you can do the following:



1. Call the **Mirror array** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).

2. Specify the **copied object**: operations, curves, points, auxiliary planes and axes.

For details about copied object selection, see Section 2.7.1.3.1 on p. 512.

The names of the selected objects will appear in the **List of Objects** toolbar.

Parameters and **Properties** tabs will appear on the Property bar.



You can also select objects for copying prior to calling the array creation command.



3. Enable the **Plane** switch and specify the **symmetry plane** in the Design Tree or mode window. The name of the selected object is displayed in a field beside the switch.

4. If required, you can also:

- ▼ Creating a **geometric array** with operations to be copied (see Section 2.7.1.3.5 on p. 515).
- ▼ Changing of array name and properties of surface of instances. Enable the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.

2.7.8.2. Creation of mirror copy of a body or a surface

To create a mirror copy of a body or a surface perform the following actions.



1. Call the **Mirror array** command.

Once the command has been called, the property bar will contain the **Choice of objects** tab (see Fig. 2.7.3 on p. 512).



2. In the **Type** group, enable the **Bodies or Surfaces** switch.

3. Specify the **copied object**: body or surface. The selected object is highlighted by a color.

For details about copied object selection, see Section 2.7.1.3.1 on p. 512.

Result of operation and **Properties** tabs will appear on the property bar.



You can also select an object for copying prior to calling the array creation command.

4. Select a **flat object** that will serve as the symmetry plane when copying the body. Such object can be any flat face and a projection or auxiliary plane.

After specifying a body (surface) and a plane of symmetry, there will be displayed the ghost of mirror copy of a body (surface).

5. Select the required **result of copying**, if a body is being copied. To do this, use the respective group of switches on the **Result of operation** bar (see Section 2.7.8.2.1 on p. 578).



6. Using switches of the **Mode** group, specify if the copied object should be maintained after execution of the command or not.



7. If required, you can also:

- ▼ Canceling the selection of all objects on the current tab of the Properties bar. To do this, click the **Reselect** button on the Advanced Control panel. Highlighting will be cleared off the previously selected object (objects), and you can select a new body.
- ▼ Changing of array name and properties of surface of instances. Enable the **Properties** tab of the Property Bar.





8. Confirm array creation

The model window will display a mirror copy of the selected body or surface, and the Design Tree will show the *Mirroring* icon.

2.7.8.2.1. Result of Mirror-Copying



- ▼ Activate the **Auto Determination** switch to have the operation results defined automatically. In this case, the construction is done as follows:

- ▼ if the mirror copy of the body does not intersect and does not have surfaces in contact with the source body, then the copy is constructed as a stand-alone body,
- ▼ if the mirror copy of the body intersects or has a common surface with the source body, then the bodies are merged into one.

The switch is active only during mirror array creation. When editing array in the first case, **New Body** switch is active on the Properties bar, in the second — **Union** switch.



- ▼ Activate the **New Body** switch to have the mirror copy as a separate body regardless of whether it intersects the source body, or not.



- ▼ Activate **Join** switch to merge the source body and its mirror copy.

When creating a new mirror copy, the result is defined automatically, i.e., by default, in the **Operation Result** group, the **Auto Determination** switch is active.



If the **Merge** switch was activated, whether manually or automatically, then a mirror copy of the body can be constructed only upon the condition that it would intersect or have a common surface with the source body. If this condition is not observed, the copy operation is marked as failing.

2.7.9. Array by Sample

You can to create an array of current assembly components located like instances of an existing array (**master array**).



You can use any type array, except for array by sample, as the master array.

Changed parameters of a master array are transferred to the array by sample and it gets rebuilt.

2.7.9.1. Creating a component array by sample

To create a component array by sample, complete the following steps.



1. Call the **Array by Sample** command.

After calling the command, the **Select Objects** tab on the Properties Toolbar will be activated.



Fig. 2.7.47. Object Selection tab

2. Select **copied components** to create the array. The names of the specified components will appear on the **List of Objects** bar.
3. Enable the **Options** tab.



- 3.1. Select the **master array**. To do this, activate the **Master Array** switch and specify the required object array in the Design Tree or in the model window.

The chosen master array will be color highlighted, and its name will be displayed near the switch.

The array ghost will be displayed.

- 3.1. Choose the **method of positioning the array's instances** with respect to the master array (see Section 2.7.9.2 on p. 579).
- 3.2. If the master array contains a nesting (i.e. its source object is another array), you can control the **number of instances** of the created array using the **Consider nesting** option.

If the option is enabled, the number of instances of the created array will be equal to a multiplication of the master array's instances number by the number of instances of the nested array. If the option is disabled, it is just the number of instances of the master array.

- 3.3. If the master array has deleted instances, you can control the **number of instances** of the created array using the **Consider deleted** options (see Section 2.7.1.3.6 on p. 516 for more information about deleting instances).

If the option is enabled, the created array will not contain instances with the same numbers as the numbers of deleted instances of the master array, otherwise, the instances will be contained in the created array.

All the changes will be reflected in the array ghost.

4. If necessary, change the array's name. To do this, use the controls of the **Properties** tab of the Property Bar.



5. Confirm array creation

Created array appears in the model window, while the icon attributable to its type is displayed in the Design Tree.

2.7.9.2. Array instances positioning

There are methods to position array's instances using a sample.

Method 1

Instances of the created array are positioned in space in accordance with the master array parameters.

In this case, data about its type and parameters are retrieved from the master array. This data is then used to create an array by a sample. For example, if an array by grid is used as the master array, data about array axes' objects and instance step value for each axis of the grid are retrieved. The result of the array creation will be the same as when creating a component array of the same type as the master array using the same parameters.

In Fig. 2.7.48, you can see example of creating an array by sample with parameters of the sample array. The array by grid is used as a sample. Its instances are shown in gray color. The copied object of each array is located in its lower left corner (highlighted by thick lines).

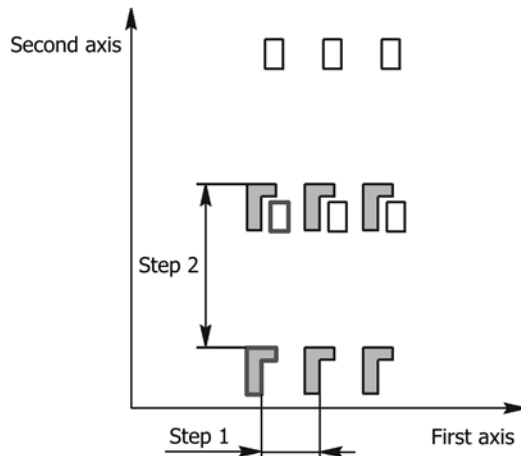


Fig. 2.7.48. Locating instances of the array by sample in accordance with parameters of the sample array



In order to create an array by sample using this method, enable the **By parameters of master array** switch in the **Method** group on the property bar.

Method 2

Instances of the array to be created are positioned against their respective instances of the master array in the same way as the object to be copied is positioned against its respective instance of the master array (base instance). By default, the nearest to the copied component instance is selected as a base instance in the master array. If several components are copied, then the instance closest to the component, that had been specified first, is selected as the base instance in the sample array. If necessary, you can select another base instance.

In order to create an array by sample using this method, do the following.



1. In the **Method** group on the Property Bar, enable the **Save position with respect to the instance of the sample array** switch.

The base instance of the master array, which is chosen by default, will be color highlighted. The array's ghost will be rebuilt — array's instances are positioned against their respective instances of the master array in the same way as the copied component is positioned against the base instance of the master array. The **Instance** control element will appear on the property bar.



2. In order to change the base instance, enable the **Instance** switch and specify the required instance of the master array in the Model Tree or window. The name of the selected instance is displayed in a field next to the switch. The array's ghost is being reconstructed.

Example of creating an array by sample using this method can be found in Fig. 2.7.49. The array by grid is used as a sample. Its instances are shown in light gray color. The copied component of the created array is shown in dark gray color. The base instance of the sample array is highlighted using thick lines.

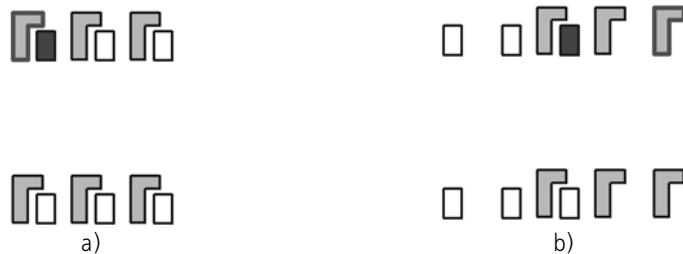


Fig. 2.7.49. Locating of instances of array by sample keeping mutual disposition of the copied component and the base instance of the sample array:
 a) base instance of the sample array is selected by default;
 b) base instance of the sample array is specified by the user

2.7.10. Array of arbitrary objects

An option to create an array of arbitrary set of objects is available for all arrays with the exception of mirrored arrays and the arrays by sample. This variant makes it possible to create an array made of different types of objects within a single operation.



In order to create an array of arbitrary objects, enable the **Arbitrary** switch in the **Type** group after you call the command to create the required type of array (for example, array by grid). The following tabs will appear on the Properties bar: **Parameters**, **Instances**, **Operation result** and **Properties**.

All steps related to creation of an array of arbitrary objects are the same as steps related to creation of an array of the selected type of objects (for example, operations). The following are exceptions:

- ▼ object selection for copying (see Section 2.7.10.1 on p. 581),
- ▼ setting the array's application area (see Section 2.7.10.2 on p. 585).

Note the following features when exploding an array of arbitrary objects. Once an array is exploded, all objects of the array are transformed into objects that are absolutely the same as the source ones. Exceptions are sketches. They will no longer have any link to their base planes after explosion of the array. Such sketches are located in the model using the **Independent location** method (see Section 2.1.9.1 on p. 141 for more information about independent location).

2.7.10.1. Main principles of object selection

When creating an array of arbitrary objects, various types of objects can be selected for copying — operations, surfaces, curves, points, auxiliary planes, axes, sketches, coordinate systems and so on.

It is impossible to select bodies, components, previously created arrays and some operations (ribs, shell, sheet body, thicken and so on) for copying.

Objects to be copied are selected in the Design Tree. The names of the selected objects will appear in the **List of Objects** toolbar.

If the **Select intermediate elements** option is enabled, the set includes not only selected objects but also the objects located between them in the Design Tree. The inclusion of intermediary objects to the set does not depend on the Tree view you are using — sequence of construction or model structure. The first, the last, and intermediate objects are defined based on the sequence of model building. Objects that cannot be copied are not included in the set.

The following principles should be followed when selecting objects in order to correctly create an array.

1. When creating an array of arbitrary objects, the same constructions are created for each instance as for the object being copied. Therefore, in addition to the objects for copying, other objects required for their construction should be specified.

Examples:

▼ **Example 1.** A sketch was used to create the extrusion operation. When selecting objects for copying, both the extrusion operation and its sketch need to be specified. Each instance of the array will include the sketch copy and the extrusion operation copy. Otherwise, all instances of the array will be created in one sketch (source extrusion operation sketch) and will coincide with it.

▼ **Example 2.** 3D curves (segment and spline constructed by previously created points) are the axis and section of the revolution surface. When selecting objects for copying, the following objects should be specified: revolution surface, segment, spline and points to be used for spline construction. Each array instance will include copies of all of these objects.

If all objects, apart from the segment (revolution axis), are specified, all instances will be created around the revolution axis of the surface being copied.

If only the revolution surface, segment, and spline are specified, the spline of the copied surface will be used for construction of all instances, since spline copies in instances cannot be created if no source points are specified and so on.

2. In some cases, when creating an array, it is impossible to specify objects required for construction of the copied object (see Example 1) or it may be difficult to do so (see Example 2). In order to create array instances correctly, links between the copied object and its source objects should be removed. For this purpose, use the **Isolate objects** command described in Section 2.7.10.1.1 on p. 584. Once the command is applied, the *Copying* object is created. It should be specified together with copied object during array creation.



If a sketch is one of the source objects, note the following. The sketch is not included in the copy resulting from object isolation. The copied object maintains the link to the sketch. Therefore, the sketch needs to be specified when creating an array.

Examples:

▼ **Example 1.** A hole is built on the face resulting from sketch extrusion. Edges of the face define the hole's position. In order to create an array of holes, it is necessary to specify the hole for copying and its source extrusion operation. Each instance of the array will

contain both the hole and the extrusion operation. This will result in modified parameters of the body. To avoid this, you need to specify not the whole extrusion operation but only those of its objects that are absolutely required to create the hole, i.e. flat face on which the whole is located and edges defining its position. Apply the **Isolate objects** command to the hole. As a result, a copy will be created that includes only those objects that are required for the construction. The link between the hole and the extrusion operation is replaced by the link to the resulting copy. Now, when creating an array, you need to specify the copied hole and the copy of its source objects.

- ▼ **Example 2.** The surface is created by grid. In order to correctly create instances of this surface in an array, you need to specify both the surface and all of its points. This may take a long time. If you apply the **Isolate objects** command to a surface, you will need to specify only the copied surface and the *Copying* object resulting from its isolation when creating the array.
3. A sketch may be one of the source objects of the copied object. If the sketch is linked to a CS starting point (for example, by means of a controlling dimension), you will need to specify both the sketch and its CS for the array to be created correctly. CS specification may also be necessary if you need to specify coordinates of objects in instances when creating an array and handling it later.

If CS is specified, a copy of CS will be created for each instance with respect to which coordinates of objects included in the instance are determined. Values of the coordinates cannot be changed. Otherwise, no CS copies are created. Coordinates of objects included in the instances are determined relative to the source CS and have various values.

Examples:

- ▼ **Example 1.** A sketch was used to create the extrusion operation. It represents a circle with the center linked to the starting point of absolute CS by controlling dimension. When selecting objects for copying, the extrusion operation, its sketch, and its absolute CS need to be specified. Each array instance will include copies of all of these objects: extrusion operation, sketch with dimension, and CS. The dimension is entered between the center of the circle in the instance's sketch and the starting point of CS related to the instance. If CS is not specified during the array creation, its copies will not be created in instances. Circles in instances' sketches remain linked to the source CS. Therefore, copies of the circle coincide with the source circle, while copies of the extrusion operation coincide with the source operation.
- ▼ **Example 2.** Coordinates of the segment's ends are set in absolute CS. If a segment and absolute CS are specified during object selection, each array instance includes a copy of the segment and a copy of CS. Coordinates of the segment's ends in the instance are determined with respect to CS of the instance, therefore, values of the coordinates remain unchanged. If only a segment is specified during the array creation, copies of CS will not be created in instances. Coordinates of segment's ends in the instances are determined relative to the source CS and have various values.



Ghosts of CS copies are not displayed when creating an array of arbitrary objects.

2.7.10.1.1. Object isolation

In some cases, it is necessary to specify only some source objects of the object being copied (not all of them) in order to create an array correctly. For this purpose, object isolation is used. Examples of object isolation can be found in p. 2 of Section 2.7.10.1 on p. 581.

In order to isolate an object from source objects, select it in the Design Tree and call the following command: **Editor — Isolate objects** (if a body or surface is specified, the command is not available).



It is recommended to specify objects to be isolated when displaying the model construction sequence in the Tree.

Once the command is called, a copy of source objects of the object being isolated appears in the model. The copy is a model object that contains geometric objects without history. These objects are copies of the source geometric objects of the object being copied and are connected with them by associative links.



The copy is created in absolute coordinate system and is displayed in the model creation tree as the *Copying* object which is located in the Tree in front of the isolated object (if the display of model construction sequence is enabled). Objects included in the copy are displayed in the window of the model. If necessary, you can hide objects of the copy.

As a result of the command execution, links between the isolated object and its source objects are removed, while links between the isolated object and objects of the resulting copy are created.

The copy includes surfaces, faces, vertices, and edges (depending on the set of source objects).

For example, a surface results from rotation of a circle's arc about the edge of an extrusion element. When isolating a revolution surface, a copy is created which includes a circular arc (section of rotation) and an edge (axis of revolution). Operation of extrusion, which includes the edge, is not included in the copy.

If the source object is a sketch, the copy includes only those sketch objects that serve as an axis, path, guiding object. Other objects and the sketch itself are not included in the copy. The isolated object remains linked to the sketch.

When only a sketch and/or coordinate system are the source objects of an object to be isolated, the object cannot be isolated. Once the command is called, a message is displayed on the screen saying that the object has no replaceable links. Exceptions exist when a sketch contains an object which is an axis, path, or an object that defines building direction. In such cases, the object is isolated.

You can isolate several objects at once, if they are located one after each other in the Tree (if the model construction sequence display is enabled). For this purpose, select the required objects prior to calling the command using the <Shift> or <Ctrl> key.



It is impossible to isolate arrays, layout elements, and some operations, such as chamfer, fillet, slope, shell, boolean operation, change position, deduct components, copying, scaling, or sheet body operations.

2.7.10.2. Application area of the array of arbitrary objects

You can set the operation's application area when creating an array of arbitrary objects. For general information about application area of an operation, see Section 2.3.10.1 on p. 217.

In order to set an array's application area, use the **Operation result** tab in the Properties bar (Fig. 2.7.50).

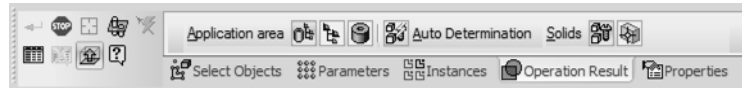


Fig. 2.7.50. Setting an array's application area

This tab contains the **Application area** group of switches where the type of an operation's application area can be selected:



- ▼ Components and bodies – this operation is applied to components included in the model and to bodies created in the model,



- ▼ Components – this operation is applied only to components included in the model,



- ▼ Bodies – this operation is applied only to bodies created in the model.



Once the application area is selected, specify objects that make up the area. In order to specify bodies, elements in the **Operation result** tab are used. If components should be included in the application area, lick the **Components application area** button in the Special Control Panel. After this, the required controls will appear in the **Parameters** tab (see Section 2.3.10.4.1 on p. 222).



By default, application area is determined automatically. The **Auto determination** switch will be enabled both for bodies and components.

Auto determination means that the application area includes all objects (bodies, components, or both) intersected by the array's instances, except for objects that are hidden or excluded from calculation.

If an array's instance contains an operation of adding a material, it is joined with the body (if there is an intersection with a component, the instance is joined with the component's body). If there are no intersections, the instance is created as an independent body.

If an array's instance contains an operation that deletes material, an error will occur if there are no intersections in the model.

A specific feature of auto determination is formed when working with an array is that no list of objects making up the application area. If the model's composition is modified, the array application area will also be changed automatically.



Automatically determining an array's application area is preferable to manually specifying objects. The system will determine on its own which bodies and components each array's instance intersects with and includes the objects in the application area.

If bodies are specified manually, note that each specified body should intersect all instances of the array. Otherwise, an error will occur in the model.

In some cases, you need to manually select objects that make up the application area. For example, faces of two bodies adjoin each other. There is a hole, created in one of the bodies, the depth of which is defined by the **Through all** method. If the application area is determined automatically when creating an array, the resulting instances of holes will go through both bodies. To avoid this, the required body should be specified manually.

Manual specification of bodies is described in this section. For components, see Section 2.3.10.4.2 on p. 224.

In order to specify the required bodies, disable the **Auto determination** switch. The **Bodies** group will appear in the Properties bar. The following switches will be available there:



- ▼ **All bodies** makes it possible to include all bodies in the array's application area, except for hidden and excluded from calculation, regardless of whether they intersect with the edited feature or not.



- ▼ **Body Selection** allows the bodies that will be included in the current operation application area to be selected.

Upon activating the **Body selection** switch, the **List of Bodies** toolbar becomes available. It contains the list of bodies included in the operation application area and two buttons: **Select All** and **Delete**.

To add a body to the application area, select it in the Design Tree or in the model window.



The **Select All** button serves to include all bodies present in the model in the operation application area, including hidden bodies and bodies that do not intersect with the feature being edited.



The **Delete** button serves to exclude a body from the application area.



An array of holes with chamfers and fillets can be created correctly only within the body (component) that contains the objects to be copied. If an array's instance intersects another body (component), no chamfer (fillet) will be created. An error will occur in the model.

2.8. Auxiliary Items

2.8.1. Auxiliary Axes

If the existing edges in the model are not enough for object creation, you can create auxiliary axes.

You can find the auxiliary axis creation commands in the **Operations** menu, and the corresponding command calling buttons are grouped together on the **Auxiliary Geometry** toolbar.

Each auxiliary axis command will prompt you to specify the reference objects of the axis being created. For the purposes of construction, mostly **point**, **straight-line**, and **flat** model objects are used (they are listed in Table 2.1.7 on p. 131).



If objects were selected before calling the command, they will be used as the base objects for axis creation.



The created axis will be displayed in the Model window as a line segment. The Design Tree will show a special icon.

An axis-depicting line segment slightly protrudes beyond the extents of the objects selected to construct that axis. Sometimes, to better understand an axis location, you may want its rendering line segment to be longer (shorter) or be located at another part of the axis (straight line). You can change the size and position of this line segment by dragging its handles with the mouse (those appear when the axis is selected).

2.8.1.1. Axis through Two Vertices



To create an auxiliary axis going through two specified point objects, call the **Axis through Two Vertices** command.

Specify a pair of point objects that the axes will go through.

2.8.1.2. Axis on Plane Intersection



To create an auxiliary axis on the intersection line of two flat objects, call the **Axis on Plane Intersection** command.

Select two flat objects to create an axis at their intersection.

2.8.1.3. Axis through Edge



To create an auxiliary axis going through a straight-line object, use **Axis through Edge** command.

Specify the straight-line object that the new axis will go through.

2.8.1.4. Conical Surface Axis



Call the **Conical Surface Axis** command to create the auxiliary axis which is a revolution surface axis.

Specify the revolution surface which axis must be created.

2.8.1.5. Axis through Vertex along Object

You can create the auxiliary axis passing through a point object in the direction defined by a selected object or vector.

2.8.1.5.1. Objects Used for Construction

In order to create an axis, a point object and a guiding object are required.

Any of the objects listed in Table 2.8.1 or a vector created during the process of axis construction can be used as a guide object.

Tab. 2.8.1. Guide Objects

Object	Object-Defined Direction
Straight object	A straight line parallel or perpendicular to an object ^{***} .
Planar object	A straight line perpendicular to an object.
Flat curve:	A straight line perpendicular to a curve plane.
▼ arc,	
▼ circle,	
▼ a sketch line.	
3D curve [*]	A straight line directed along the tangent vector or vector of the principal normal ^{***} .
Revolution surfaces ^{**}	A straight line parallel or perpendicular to the revolution axis ^{***} .
Free-form surfaces [*]	A straight line perpendicular to an object at the indicated point.

* The point object must belong to a guide object.

** You can use sphere as a guide object in case the point object belongs to it. The direction defined by the sphere is a perpendicular line at the specified point.

*** Use the **Orientation** group of switches to select the required direction of the axis being created (see Section 2.8.1.5.2 on p. 588).

2.8.1.5.2. Axis creation



Call the **Axis through Vertex along Object** command to create the auxiliary axis passing through a vertex in the direction of a selected object.

If objects already exist in the model, specify them.



When you specify the point drawn by the **On Curve** (see Section 2.5.2.5 on p. 338) or **On Surface** (see Section 2.5.2.6 on p. 339) method, there will be automatically selected a guide object for the axis - a curve or surface on which the point is created. If necessary, you can select some other guiding object. For this, specify the required object in the model window.

Objects for constructing an axis can be also selected before calling the command.

If the model does not contain the required objects, you can create them without exiting the command.



- ▼ To create a point, press the **Point Creation** button on the Special Control Panel (see Section 2.5.2.11 on p. 346).



- ▼ To create a vector defining the axis direction, press the **Vector Drawing** button on the Special Control Panel (see Chapter 2.8.5 on p. 613).

Upon creating an object the system resumes the axis construction process.



Note that the object selected in the list is highlighted in the Design Tree. Change of the point parameters is only possible in the object editing mode rather than in the axis editing mode.

The ghost of the axis being created will be displayed once the objects are selected.

In some cases (see Table 2.8.1 on p. 588), you can select the direction of the axis being created: along or perpendicular to the guiding object. This is done by the **Orientation** group of switches.



- ▼ **Parallel to object,**



- ▼ **Perpendicular to object.**

Change of axis orientation is shown on the arc ghost in the model window.



To change objects used for axis construction, press the **Select Other** button on the Special Control Panel and re-select objects.

You can specify axis name and color in the **Properties** tab of the Property Bar.



To finalize the construction, click the **Create Object** button on the Special Control Panel.

2.8.2. Auxiliary Planes

If the coordinate planes and the flat facets, existing in the model, are not sufficient for object creation, you can create auxiliary planes.

You can find the auxiliary plane creation commands in the **Operations** menu, and the corresponding command calling buttons are grouped together on the **Auxiliary Geometry** toolbar.

Each auxiliary plane creation command will prompt you to specify the base objects for the plane and set its parameters in the corresponding fields on the Properties toolbar. For the purposes of construction, mostly **point**, **straight-line**, and **flat** model objects are used (they are listed in Table 2.1.7 on p. 131).



If objects were selected before calling the command, they will be used as the base objects for plane creation.

The plane with defined parameters is displayed on the screen as a ghost.



To lock the plane in the model, click the **Create Object** button on the Special Control panel.

All the commands to draw an auxiliary plane, except for the **Plane Tangent to Face at Point** command, have the auto-creation mode enabled by default. For more information on automatic and manual creation of objects refer to Section 1.4.2.10 on p. 93.



The created plane will be displayed in the Model window as a rectangle. The Design Tree will show a special icon.

A plane-depicting rectangle slightly protrudes beyond the extents of the objects, which were used as the base for constructing that plane. Sometimes, to better understand a plane location, you may want the plane-rendering rectangle to be larger (smaller) or be located at another part of the plane. You can change the size and position of this rectangle by dragging its handles with the mouse (those appear when the plane is selected).

2.8.2.1. Offset Plane



To create an auxiliary plane located at a certain distance from the selected planar object, call the **Offset Plane** command.



In the **Distance** field, enter the distance between the flat object and the plane being created.



Specify the flat object with respect to which the new plane's offset is assigned.

To specify the side of the existing plane or facet relative to which the new plane will be created, activate the **Forward Direction** or **Reverse Direction** switch in the **Offset Direction** group.



The offset distance and direction can be set using a handle (see Chapter 2.1.4.2.2 on p. 138).

2.8.2.2. Plane through Three Vertices



To create an auxiliary plane going through three point objects, call the **Plane through Three Vertices** command.

Specify the three points that the new plane will go through.

2.8.2.3. Plane at Angle to Another Plane



To create an auxiliary plane going through a straight-line object at an angle to an existing planar object, call the **Plane at Angle to Another Plane** command.



A straight object must be parallel to a planar object or lie on such an object.

Specify a flat object to which the new plane should be inclined at the given angle.

Specify the straight-line object that the plane will go through.

Input the angle between the planar object and the plane you are creating into the **Angle** field or select it in the list that drops down.



To specify the direction that the selected angle should be drawn in (relative to the planar object), activate the **Forward Direction** or **Reverse Direction** switch in the **Slope Direction** group.



The angle value and direction can be set using a handle (see Chapter 2.1.4.2.2 on p. 138).

2.8.2.4. Plane through Edge and Vertex



To create an auxiliary plane going through a straight-line object and a point object, call the **Plane through Edge and Vertex** command.

Specify the objects to construct the plane.

2.8.2.5. Plane through Plane Curve



Call the **Plane through Plane Curve** command to create an auxiliary plane passing through the specified plane curve.

The following may be used as a curve:

- ▼ a sketch line,
- ▼ spline or polyline with three vertices,
- ▼ arc,
- ▼ other flat curves: circular edge, etc.

Select the curve through which the new plane must go.

2.8.2.6. Plane through Vertex and Parallel to Another Plane



To create an auxiliary plane going through a point object and parallel to a planar object, call the **Plane through Vertex Parallel to Another Plane** command.

Select a point object and a planar object.

2.8.2.7. Plane through Vertex and Perpendicular to Edge



To create an auxiliary plane passing through the point object perpendicular to a straight object or vector, call the **Plane through Vertex and Perpendicular to Edge** command.



The point object does not have to necessarily belong to a curve.

If a point object and a straight object already exist in the model, specify them.

If the model does not contain the required objects, you can create them without exiting the command.



▼ To create a point, press the **Point Creation** button on the Special Control Panel (see Section 2.5.2.11 on p. 346).



▼ To create a vector, click the **Create Vector** button on the Special Control Panel (see Chapter 2.8.5 on p. 613).

Upon creating an object the system resumes the plane construction process.



Note that the object selected in the list is highlighted in the Design Tree. Change of the point parameters is only possible in the object editing mode rather than in the plane editing mode.

2.8.2.8. Normal Plane



To create an auxiliary plane normal to a revolution surface (except for a sphere), call the **Normal Plane** command.

Select a method to create a new plane using switches of the **Method** group on the Property Bar:



▼ **Autocreation** - to create the normal plane passing through the revolution surface axis and a point of this surface with zero values of U and V parameters (for face parametric representation refer to Section 2.5.1.3 on p. 328);



▼ **Parallel to the object**— construction of a normal plane which passes through the revolution surface axis in parallel to the specified coordinate plane or plane face.

Specify the objects to be constructed.

▼ If **Auto construction** is selected, specify the revolution surface.

▼ In case the **Parallel to Object** method is selected, specify a revolution surface and then the coordinate plane or flat face in relation to which the new plane will be located.

By default, the Object Auto Create option is enabled (see Section 1.4.2.10 on p. 93). This mode will automatically create a new plane (i.e. confirmation of the creation is not required) in the default position.



If you want to specify the new plane angle of rotation about the revolution surface axis, disable autocreation of objects (release the **Autocreation of Object** button on the Special Control Panel).

The angle will be calculated from the default position of the new plane.



Enter the required rotation angle value in the **Angle** field.

To specify the direction of drawing of the selected angle relative to the default position, activate the **Forward Direction** or **Reverse Direction** switch in the **Angle Direction** group.



The angle value and counting direction can be also set using a handle (see Chapter 2.1.4.2.2 on p. 138).



If you want to change the objects used to draw a plane, press the **Reselect** button on the Special Control Panel and then reselect the objects.



Confirm plane creation by pressing the **Create Object** button on the Special Control Panel.

2.8.2.9. Tangent Plane



To create an auxiliary plane tangential to a cylindrical or conical facet of the model, call the **Tangent Plane** command.

The initial position of the tangent plane is defined by the line of tangency. The line of tangency, in its turn, is determined as the intersection of said cylindrical or conical face and a flat object passing through its axis. The flat object must be created in advance. If necessary, the tangent plane can be rotated around the axis of the cylindrical or conical surface with respect to the initial position.

Indicate a cylindrical or conical face to which the new plane must be tangent, and then the flat object passing through its axis.

The tangent plane ghost appears on screen.

Enter the rotation angle of the tangent plane with respect to its initial position in the **Angle** field on the **Options** tab of the Properties bar. By default, the angle is equal to zero, and the new plane becomes perpendicular to the selected flat object.



The angle value can be set using a handle (handles are detailed in Chapter 2.1.4.2.2 on p. 138).



To specify the side (relative to the conical facet) on which the new plane will be created, activate the **Position 1** or **Position 2** switch in the **Plane Position** group.

2.8.2.10. Plane Tangent to Face at Point



Call the **Plane Tangent to Face at Point** command to create the auxiliary plane tangent to the specified face at the set point.

Select the face to make the new plane tangent to.

On the selected face, the ghost of the plane's theoretical surface will be displayed as a grid of the U and V isoparametric curves and the ghost of the plane being created as a rectangle (refer to Section 2.5.1.3 on p. 328 on the theoretical surface). By default, the new plane goes through the point of face specification.

Specify position of a new plane. This can be completed by the following methods:

- ▼ Specify the position of the point through which the plane should pass. The point position is determined by the offset along isoparametric curves U and V.

To specify the point offset, enter the respective values to the **Parameter U, %** and **Parameter V, %** fields.

- ▼ Associate the point (through which the plane will pass) with the existing point object (see Table 2.1.7 on p. 131). To do this, select the desired object in the Design Tree or in the model window. The **Point** button will be ticked, and the field next to the button will display the object name. This indicates the associative relation of the plane with the selected object. Due to this relation, the created plane will follow the object in case it changes its position.



If necessary, the relation can be disabled by pressing the **Point** button. The button will be unchecked, the field will be cleared.



In order to specify the plane position you can specify point objects both belonging and not belonging to the selected face, but being projected on this face. In this case, the projection determines the position of the plane being created.

If a model does not contain the required point object to construct a plane, you can create a point without exiting the command. To do this, press the **Point Creation** button on the Special Control Panel (see Section 2.5.2.11 on p. 346).

After the point is created, the system will proceed with creating the plane.



Note that the object selected in the list is highlighted in the Design Tree. Change of the point parameters is only possible in the object editing mode rather than in the plane editing mode.

2.8.2.11. Plane through Edge Parallel/Perpendicular to Another Edge



To create an auxiliary plane going through selected straight-line objects parallel or perpendicular to other straight-line objects, call the **Axis through Edge Parallel/Perpendicular to Another Edge** command.

Specify the straight-line object that the new plane will go through.



To select whether the plane will be **Parallel** or **Perpendicular** to the other straight-line object, activate the respective switch in the **Plane Position** group.



Specify the straight-line object the new plane will be parallel/perpendicular to.

2.8.2.12. Plane through Edge Parallel/Perpendicular to Face Command



To create an auxiliary plane through the selected straight-line objects, parallel or perpendicular to planar objects, call the **Plane through Edge Parallel/Perpendicular to Face** command.

Specify the straight-line object that the new plane will go through.



To select whether the plane will be **Parallel** or **Perpendicular** to the other object, activate the respective switch in the **Plane Position** group.



Select an object that the plane will be parallel/perpendicular to.

2.8.2.13. Median Plane



To create a bisecting plane of a dihedral angle, call the **Median Plane** command.

A dihedral angle is a space sector bounded by two half-planes with a common border line. These half-planes are called the **faces** of the dihedral angle, and the plane intersection line is called the **edge** of the dihedral angle. The angle between the facet intersection lines of a dihedral angle and a plane perpendicular to the edge of the dihedral angle is called the **linear angle** of the dihedral angle.

The **bisector plane** of a dihedral angle is a plane going through the edge of this angle and the linear angle bisector.

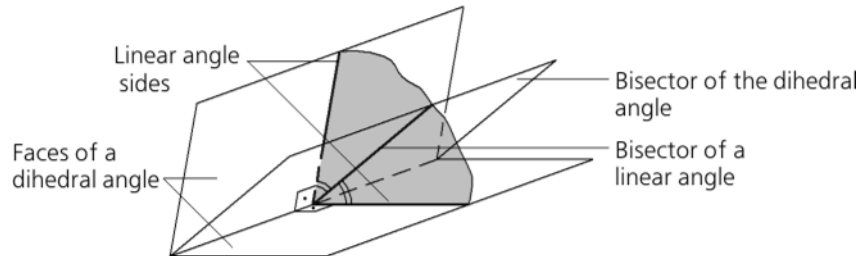


Fig. 2.8.1. Dihedral angle and its bisecting plane

A dihedral angle to construct a median plane can be defined by:

- ▼ faces; to do this, specify two flat objects,
- ▼ linear angle; to do this, select two straight objects,
- ▼ face and a side of a linear angle; to do this, select a flat and a straight objects.

To specify a median plane location relative to the selected objects, activate the corresponding switch on the **Settings** tab of the Properties toolbar:



- ▼ **Position 1:** a bisector plane will be created



- ▼ **Position 2:** a plane will be created perpendicular to the bisector plane and going through the edge of the dihedral angle.

In the special case when the specified objects are parallel, proceed with the construction in the following way (regardless of what position switch is active).

- ▼ If the objects are straight, then the median plane is constructed perpendicular to the plane going through them and at equal distances from each of them.
- ▼ If the objects are planar (flat), or if one is planar and another one is straight, then the median plane is constructed parallel to them at equal distances from each of them.

2.8.3. Coordinate systems

2.8.3.1. Absolute coordinate system

Every model has an absolute coordinate system and its corresponding planes. The names of coordinate axes and planes will appear in the Design Tree immediately after creating the new model file.

The absolute coordinate system of the model is shown in the middle of the window as three orthogonal lines in red, blue and green. All the three lines have a common origin coinciding with the origin of model coordinates, i.e. the point with coordinates 0, 0, 0.

The planes are represented by special symbols, i.e. the red, blue and green rectangles lying in their respective planes. By default the rectangles are positioned in such a way that their centers coincide with the origin of coordinates in order to visualize the spatial plane layout for the user.

In some cases the plane rectangle may need to be enlarged (reduced) or located at elsewhere on the plane to facilitate understanding of the plane position. You can change the size and position of this rectangle by dragging its handles with the mouse (those appear when the plane is selected).

The absolute coordinate system axes and planes cannot be removed from the part file. However, they can be renamed (see Section 2.1.2.5 on p. 111) as well as enabled or disabled in the model view window (see Section 2.14.1.1 on p. 781).

In the left bottom corner of the window document, you can see the model orientation control. It consists of three thick arrows — red, green and blue, indicating positive directions of the X-axis, Y-axis, and Z-axis respectively. This control allows you to change orientation of the model, e.g. by clicking the arrow or by rotating it. Working with the dialog is described in Section 2.1.3.3 on p. 124.

This feature rotates together with the mode, just like the symbol at the origin of coordinates; however, unlike the latter, it does not shift when shifting the model and cannot be disabled.



If necessary, you may create local coordinate systems in the model. For detailed information on creation and use of local coordinate systems see below.

2.8.3.2. Local Coordinate Systems

Sometimes, an absolute coordinate system fails to provide an adequate description of a part segment or it can be used only upon performing additional calculations. Usually this happens when constructing part segments which are rotated relative to the absolute coordinate system.

In these cases, to simplify work you can start by creating a local coordinate system (LCS), with its origin and axes placed as required, where you can create objects afterwards. For example, if you want to produce a polyline with its segments perpendicular to each other, but at an angle to the absolute coordinate system axes, you can first create LCS with its axes parallel to the segments of the future polyline and its origin coinciding with the polyline's first vertex.

LCS may be also used for positioning of base parts, imported surfaces, assembly components. LCS creation is described in Sections 2.8.3.5–2.8.3.7, creation of objects using LCS — in Section 2.8.3.4.

You can build several LCSs in a model, with the possibility of switching between them making currently required LCS as **the current LCS** (see Section 2.8.3.3). The coordinate system which was the current one at the moment of object creation becomes the coordinate system of this object.

Objects are constantly connected with their coordinate systems. This makes it possible to quickly change the position of the part segment by changing the position of its coordinate system.

Objects can be transferred from one coordinate system to another (See Section 2.8.3.4.2).

All LCSs available in a model are displayed in the Design Tree, similar to other objects. If model structure display is enabled in the Tree (See Section 2.1.2.2 on p. 106), then LCSs are in the *Coordinate systems* Section.

In the model window, you can disable display of individual LCSs or all coordinate systems at once, and you can also exclude LCS from calculation using the same techniques as those used for other objects (see Sections 2.14.1.1 on p. 781 and 2.14.1.2 on p. 783).



If the LCS is excluded from calculation, all objects built in it will be also excluded, and in case of removal — removed.



Standard drawing views of a model are defined based on its absolute coordinate system.

2.8.3.3. Current coordinate system. Selection of the current coordinate system

At any time, only one coordinate system of a model can be the current system.

The letter T in parentheses is displayed before the name of the current coordinate system in the Design Tree (see Fig. 2.8.2).

Current LCS is displayed in the model window in colors preset for it's elements, whereas all the rest LCSs - in gray color. The absolute coordinate system of a model is colored independently of whether it is the current one or not.

To assign a local or absolute coordinate system the “Current” status, do the following:



1. Call the **Editor— Select Current CS** command. The coordinate system which is the current one at the moment shall be highlighted in the model window.
2. Specify the coordinate system in the Design Tree that should be the current one. You can also set the origin of this system in the model window.

To assign and unassign the Current status, you can use the **Current CS** command in the context menu of the coordinate system in the Design Tree:

- ▼ If the command is ticked, then the coordinate system is already the current one; upon calling the command it shall not be the current system any longer; instead, the absolute coordinate system shall become the current one;
- ▼ If the command is not ticked, it means that the coordinate system is not the current one; upon calling the command it shall become the current coordinate system.

If you enabled the **When creating LCS, Assign it As the Current CS** option when customizing the LCS (See Section 9.1.11.9 on p. 1734), then every new LCS shall automatically gain the “Current” status.

2.8.3.4. Using LCS

- ▼ LCS may be used as the object coordinate system (See Section 2.8.3.4.1) for the objects, the position and/or orientation of which are defined in relation to the coordinate system (and not in relation to the other objects of the model). If LCS is the object coordinate system, this object can be moved and rotated in the absolute coordinate system by changing the LCS position. To make an object independent from LCS, transfer it to the absolute coordinate system of the

model (see Section 2.8.3.4.2 on p. 600).

- ▼ You can use the LCS elements, such as the origin, coordinate planes and axes, for construction of a model similar to the use of other point, planar and linear objects.
- ▼ The *Coincidence* mating imposed on two LCSs makes them (and, hence, the components that contain them) immobile with respect to each other.

2.8.3.4.1. Object coordinate system

Object coordinate system is a coordinate system with respect to which object position and/or orientation are defined.

The list of objects that use coordinate system is presented in Table 2.8.2.

Tab. 2.8.2. Objects that use coordinate system

Object	Object parameters that depend on coordinate system
Point	<ul style="list-style-type: none"> ▼ Coordinates of a point in case of using the By Coordinates (XYZ), By Cylinder (RAZ), By Sphere (RAB) techniques ▼ Axes orientation to define coordinates of a point with respect to the reference point when using the Transfer technique.
Polyline, spline	<ul style="list-style-type: none"> ▼ Vertex coordinates when using the By Points technique ▼ Axes orientation when using the By X Axis, By Y Axis, By Z Axis techniques
LCS*	<ul style="list-style-type: none"> ▼ Coordinates of the origin (LCS creation by the basic technique) ▼ Euler angles (LCS creation by supplementary techniques)
Circular arc, segment	Coordinates of points
Sketch	<ul style="list-style-type: none"> ▼ Default position of the origin ** ▼ Default axes orientation **
Imported Surface	Position and orientation.
A group of points by a curve, by a surface	Point coordinates (available after ungrouping).
Group of points from a file	Point positions. Points are created in the current coordinate system according to the coordinates saved to file.
A surface by a point grid, by a point layer	Coordinates of points (created using the Point Creation command) or position of points (retrieved from file).
Curve by law	Curve position. The curve is built in the current coordinate system in accordance with the predefined functions.

Tab. 2.8.2. Objects that use coordinate system

Object	Object parameters that depend on coordinate system
Body or surface position change operation	Body or surface position. A body or surface is linked to a hypothetical coordinate system, the origin of which coincides with the current coordinate system; in order to define the position of the body/surface, the required position of the hypothetical coordinate system is defined.
Pattern by Table	Pattern instances position. The array is built in the current coordinate system using points with coordinates saved in file.
Area	Point coordinates of the box (when creating zones by coordinates) or orientation of the dimensional box of the objects (when creating a zone by dimensions of the objects).

* LCS settings can be defined in some other LCS if the system customization (see Section 9.1.11.9 on p. 1734) has been performed with the **Create LCS in Absolute CS only** option disabled.

** When editing a model afterwards, the position and orientation of the sketch coordinate system can be changed using the **Place Sketch** command (see Section 2.14.3.1.2 on p. 787).

Spirals and round holes are connected with the coordinate system indirectly: the coordinate system is used by the sketch which is automatically created when constructing a spiral or a hole.

At first, a coordinate system of an object is the system which was the current coordinate system at the moment of object creation. How to make a coordinate system the current one, see Section 2.8.3.3 on p. 597.

If required, you can transfer an object from its coordinate system to another system (see Section 2.8.3.4.2).



Use of LCS axis, plane or origin of coordinates when creating an object (for example, creating a sketch on XOY plane of the local coordinate system) does not make LCS the coordinate system of this object.

If an object is constructed in the local coordinate system instead of the absolute coordinate system, the icon of this object in the Design Tree is marked with LCS sign which is a mini LCS icon (Fig. 2.8.2).

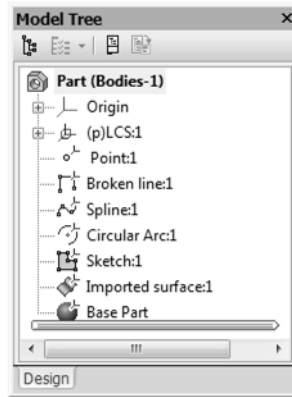
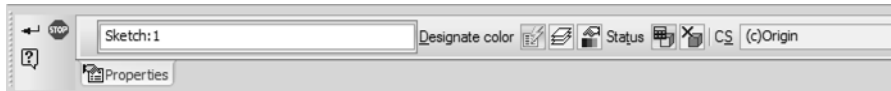


Fig. 2.8.2. Display of objects built in the LCS in the Design tree

During creation or editing of this object, the name of its coordinate system is displayed on the **Properties** tab of the Properties toolbar.

The **CS** field is also displayed on the Properties toolbar when viewing object's properties (Fig.2.8.3). To view the properties, call the **Properties** command from the context menu of the object icon in the Design Tree.

Fig. 2.8.3. **CS** field on the Property Bar

2.8.3.4.2. Replacing the object coordinate system

Any of the objects listed in Table 2.8.2 can be transferred from its coordinate system to another coordinate system.



Object transfer is only possible using the coordinate systems located above the object in the Design Tree.



To replace the object coordinate system, call the **Editor — Transfer to CS** command.

Upon calling the command you shall see, on the Properties toolbar, the controls for transferring objects to another coordinate system.

Specify the objects in the Design Tree, the coordinate system of which is required to be changed.

The names of the selected objects shall appear in the **List of Objects** toolbar (Fig.2.8.4). The object name is followed by the name of its coordinate system in parentheses.



Objects to be moved to another coordinate system can be also highlighted prior to the command calling.

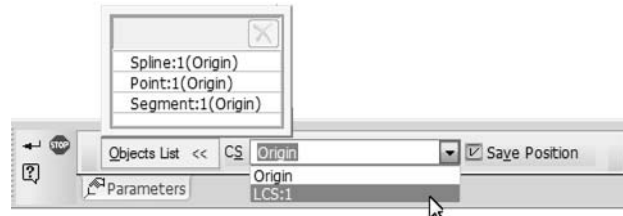


Fig. 2.8.4. Controls that allow you to transfer objects to another coordinate system



To delete any object from the list, highlight it and press the **Delete** button, located above the list.

Select the coordinate system to move objects to, by specifying it's name in the **CS** list.

The **Save position** option serves to specify if an object position should be saved in the absolute coordinate system of a model after transferring to another coordinate system.

- ▼ If the option is enabled, object parameters shall be recalculated with respect to the new coordinate system so that the object position in the model shall not change.
- ▼ If the option is disabled, object parameters shall be saved whereas the object position in the model shall change.



Having finished the setting, press the **Create object** button on the Special Control Panel to transfer objects to the selected coordinate system.

Features of object transfer from one coordinate system to another

- ▼ Parameters of some objects (polyline, base part, etc.) are not recalculated, i.e. they change their position regardless of the **Save Position** option status. Such objects are marked with a lock icon in the list.
- ▼ Elements of an object can have associative relation with other objects. In the result of replacement of the object coordinate system, the position of related elements do not change irrespective of the **Save position** option status. For example, there is an arc of circle built by three points, with two of them being defined by their coordinates and the third one coinciding with the vertex. Upon transferring this arc to another coordinate system with the **Save Position** option disabled, the points defined by coordinates shall move, whereas the point related to the vertex shall remain in it's place.
- ▼ LCSs can be transferred from one coordinate system to another only if during the LCS setting (see Section9.1.11.9 on p. 1734) the **Create LCS Only in Absolute CS** option is disabled.

2.8.3.5. LCS creation. General Information

The main settings of LCS that define it's location in space are position and orientation.

- ▼ **Position** is a setting defining the LCS center location.
- ▼ **Orientation** is a setting defining direction of LCS axes.

LCS settings are defined with respect to it's own coordinate system. Own coordinate system of LCS is the absolute coordinate system of the model or the current coordinate system (it depends on the LCS setting, see Section9.1.11.9 on p. 1734).



To start LCS creation, call the **LCS** command. The command calling button can be found on the **Auxiliary Geometry** toolbar.

Upon calling this command, the process of LCS creation by the basic technique is started. See detailed description of this process in Section 2.8.3.6.



There are also additional techniques available for LCS creation. To access them, press the **Additional Methods** button on the Special Control Panel. The process of LCS creation by additional methods shall start. See detailed description of this process in Section 2.8.3.7.

Using either the basic or additional methods, define position and orientation of the LCS being created. All updates shall be displayed on the LCS ghost in the model window.

An LCS name may be changed in the **Properties** tab of the Property bar. The name of the own coordinate system for the LCS being created is also displayed there.



Having obtained the required LCS location, fix it by pressing the **Create object** button of the Special Control Panel.



The created LCS shall be displayed in the model window, and its icon shall appear in the Design Tree.

If LCS setup (See Section 9.1.11.9 on p. 1734) is performed with the **When creating LCS, Assign it As the Current CS** option enabled, the created LCS shall become the current one.

2.8.3.6. Basic method of LCS creation



This method of LCS creation is available directly upon calling the **LCS** command. Property toolbar contains controls shown in Figure 2.8.5.

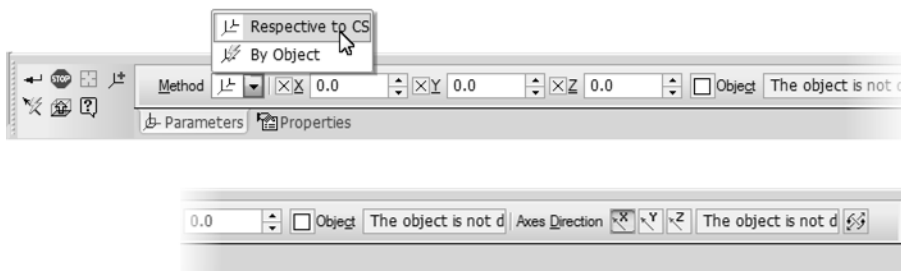


Fig. 2.8.5. Property toolbar if LCS is created by the basic method

A ghost of the LCS being created shall be displayed on the screen. By default, the new LCS coincides with its own coordinate system.

You can specify position and orientation of the LCS being created:

- ▼ with respect to the coordinate system (See Section 2.8.3.6.1),
- ▼ with respect to some object if its properties provide for unique determination of the LCS settings (See Section 2.8.3.6.2).

To select a method of determination of the LCS position and orientation, open the **Method** dropdown list and specify the required line in it (Fig. 2.8.5).

2.8.3.6.1. Creation of LCS with respect to a coordinate system

When creating LCS with respect to a coordinate system, a ghost shown in Figure 2.8.6 shall be displayed.

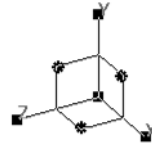


Fig. 2.8.6. LCS ghost

LCS position

To define the LCS position, enter the coordinates of its point of origin in **X**, **Y**, **Z** fields or specify a point object in the model window. Point objects used for the LCS creation are the same as those used for the point creation (see Section 2.5.2.1.1 on p. 335). The LCS ghost shall move so as to have its origin coinciding with the specified point.



To change the selected point object, press the **Define Again** button on the Special Control Panel and specify the required object in the model window.

If a point object is defined to specify the LCS position, the LCS origin shall have an associative connection with this object. Note that:



- ▼ **X**, **Y**, **Z** fields contain the object coordinates and cannot be edited,
- ▼ the **Object** button is ticked, whereas the **Object** field shows the object name.

Due to the associative connection, the LCS origin shall follow the object in case its position is changed.



To cancel the associative relation with the point object, click the **Object** button. The tick on the button will disappear.



To restore the associative relation of the point with the point object, specify the required object again.

You can move LCS in the model window with the mouse using the ghost's handles. Use the points which are displayed in the form of squares to specify the LCS position.

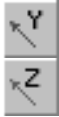
- ▼ The LCS origin point can be moved at random. The LCS ghost shall follow the handle. As this takes place, the current coordinates of the point are displayed in **X**, **Y**, **Z** fields; if one or two coordinates are fixed, displacement occurs in a plane or along an axis.
- ▼ End points of the axes can be moved in the direction of these axes. The LCS ghost will move in the same direction. The current coordinates of the LCS origin are displayed in **X**, **Y**, **Z** fields.

LCS orientation

To define the LCS orientation, enable the switch corresponding to the axis, the direction of which you want to define, in the **Axis Directions** group.



- ▼ X Axis,



▼ Y Axis,

▼ Z Axis.

The selected axis shall be highlighted on the ghost in the model window.



The axis, the direction of which should be changed, may be pointed with the mouse in the model window. For this, move the cursor to the required axis and left click the mouse when the *Guide Object of Axis* line will appear beside the cursor.

Select a guide object. When creating LCS, the same linear and planar guide objects are used as those involved in point creation (See Section 2.5.2.1.1 on p. 335).

Moreover, the guide object may be represented by a vector. In this case direction of the selected axis will coincide with the vector direction. You can create this vector using the **Vector Drawing** command (see Chapter 2.8.5 on p. 613).



The name of the selected object is displayed in the **Axis Guide Object**. The LCS axis shall have an associative connection with this object. If the vector defines the axis direction, message "Vector" appears in the field.

Direction can be set for two axes only. The third axis will be automatically directed in such a way so as to form a right-hand system with the first two.

After you select a guide object or draw a vector for the first axis, the LCS will rotate so that this axis matches the direction specified by this object or vector.



The guide object direction for the second axis can be non-perpendicular to the first axis. In this case the direction of the second axis is defined as a projection of the direction, defined by the object, on the plane perpendicular to the first axis.

If the object is either parallel to or on the same axis with the first axis, such object can not be used as a guide for the second axis.

If the vector is collinear to the first axis, such vector can not be used as a guiding object for the second axis. The message will pop up if an error take place.

If required, you can set the opposite direction for any of the LCS axes. This is done by enabling, first, the radio button corresponding to the axis in the **Axes Directions** group, then the **Reverse Axis** radio button.



You can use the mouse to reverse LCS axes. For this, having the `<Ctrl>` key pressed, left click the mouse on the characteristic point at the end of the required axis.

You can arbitrarily rotate the LCS with the mouse in the model window by moving the ghost handles. The points displayed as circles serve to define the LCS orientation. Each handle corresponds to one of the LCS planes and allows rotating the LCS around the axis perpendicular to this plane.

The axis of the LCS rotation remains immobile, whereas the other two axes are displaced. If any of them was associated with the object, this associative connection is removed.

2.8.3.6.2. LCS creation with respect to object

Initially, there is no LCS ghost in the model window when creating LCS with respect to object. It appears only upon selection of the object that defines the LCS position. The object with respect to which an LCS is created, uniquely defines its position and orientation (Table 2.8.3).

Tab. 2.8.3. Objects for LCS creation

Object	LCS settings defined by the object	
	Ref.No.	Orientation
Sketch	The sketch coordinate system origin	X coincides with the X axis of the sketch coordinate system, Y coincides with the Y axis of the sketch coordinate system Z auto determination
Intermediate vertex of a polyline	Polyline vertex	X parallel to the previous segment of a polyline, Y parallel to a projection of the following polyline segment on the plane perpendicular to the X axis, Z auto determination
A point created by the method On curve*	Point	X parallel to the tangent to the curve in the point, Y parallel to the main normal to the curve in the point Z auto determination
A point created by the method On surface or Projection	Point	X parallel to the tangent in the U direction ^{**} , Y: parallel to the tangent in the V direction ^{**} , Z auto determination
Circular Arc or Elliptical Arc (curve, contour in a sketch or an edge)	Arc center	X directed to the start vertex of the arc, Y auto determination, Z perpendicular to the arc's plane
Coordinate system	CS origin	X coincides with the X axis of the coordinate system Y coincides with the Y axis of the coordinate system Z coincides with the Z axis of the coordinate system
Body ^{***}	Center of Gravity	X coincides with the Ox axis of inertia, Y coincides with the Oy axis of inertia, Z coincides with the Oz axis of inertia.
Planar face	Center of Gravity of a face ^{****}	X coincides with the X axis of the face coordinate system, Y coincides with the Y axis of the face coordinate system, Z auto determination

- * It is impossible to use a point created on a linear curve for LCS creation.
- ** U and V directions are shown on the screen, for example, during the point construction by the **On surface** method.
- *** Bodies which have no axes of inertia (for example, a sphere) cannot be used. If the selected body has one axis of inertia (for example, a cylinder), the LCS's Z axis coincides with it.
- **** You can choose a variant of a faces' mass center position. For this purpose, the **Only outer contour** option on the Property bar is used (see below).

To define an object for LCS creation, specify it in the Design Tree or in the model window. The name of the selected object shall be displayed in the **Object** field. LCS is associated with this object.

The **Only outer contour** option controls the position of a flat face mass center which defines the LCS position. If the option is enabled, only the outer face's contour will be taken into account during the face's mass center calculation. If the option is disabled, the face's mass center is calculated taking account of the face's internal contours.

Variants of the flat face's mass center position can be seen in the figure; the mass center is marked by a circle.

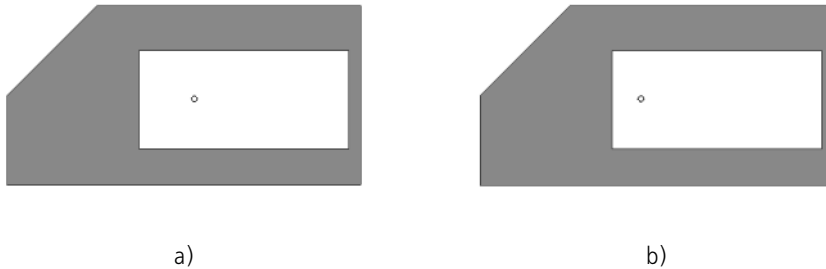


Fig. 2.8.7. Variants of a flat face's mass center positioning:
 a) if the **Only outer contour** option is enabled,
 b) if the **Only outer contour** option is disabled

2.8.3.7. Additional methods of LCS creation



To switch to additional methods of LCS creation, press the **Additional Methods** button on the Special Control Toolbar after calling the **LCS** command.

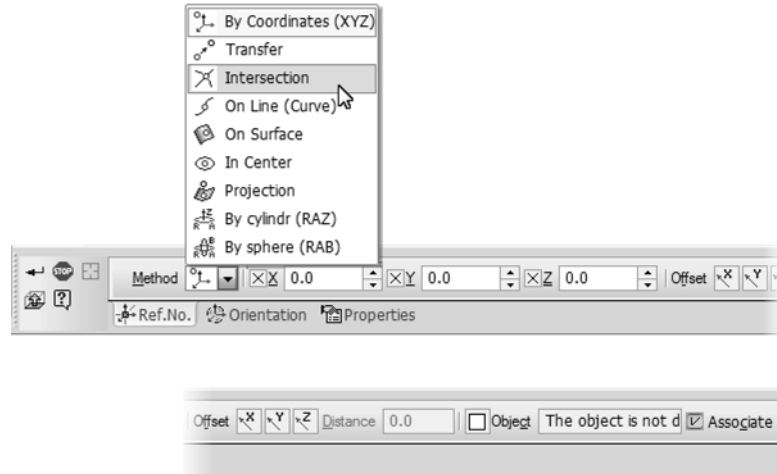


This makes the **Settings** tab on the Properties toolbar to be replaced with two tabs: **Position** and **Orientation**. They serve to specify LCS positions and orientations using various techniques. See detailed description of these tabs in sections 2.8.3.7.1 and 2.8.3.7.2.

2.8.3.7.1. LCS position

Use the **Position** tab of the Property toolbar (Fig. 2.8.8) to define the LCS position.

Controls of the **Position** tab allow you to define the position of the LCS origin using the methods available in the **Point** command (see Section 2.5.2.1 on p. 334).

Fig. 2.8.8. Position **Tab**

The only difference of the LCS positioning process from the point creation process is the presence of additional controls in the **By coordinates** method. These controls are the **Offset** group of switches and the **Distance** field. They provide for moving the LCS along its own axes.

To select an axis along which you want to displace the LCS, enable the relevant switch in the **Offset** group, then enter an offset distance in the **Distance** field. The positive value means an offset in the axis direction, the negative value – in the opposite direction.

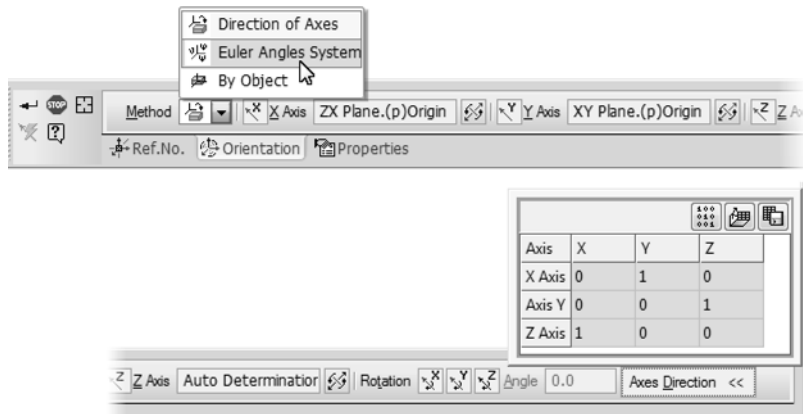


The **Distance** field is zeroed after every input, and a new LCS offset is counted from its current position.

For any changes of the LCS position its orientation remains constant.

2.8.3.7.2. LCS orientation

To define the LCS orientation, use the **Orientation** tab of the Properties toolbar (Fig.2.8.9).

Fig. 2.8.9. **Orientation** tab

Three methods of the LCS orientation are available:

- ▼ defining guide objects or drawing vectors for axes and, if required, rotating the LCS around its own axes,
- ▼ LCS rotation with respect to its own coordinate system by setting Euler angles,
- ▼ with respect to some object if its properties provide for unique determination of the LCS orientation.

To select the LCS orientation method, open the **Method** list and select the required line in it (Fig.2.8.9). See below detailed description of the orientation methods.

For any changes of the LCS orientation, its position remains constant.

When defining the LCS orientation using the **Axes Directions** and **Euler Angle System** methods, the **Axes Directions** toolbar is available on the Property toolbar. It contains a table of coordinates of unit vectors, which are co-directed with the LCS axes. These coordinates are defined in the LCS's own coordinate system. Values in the table shall change if the LCS orientation changes. It is impossible to manually edit coordinates of the unit vectors. The **Axes Directions** toolbar contains three buttons.



- ▼ **Initial Orientation** Allows cancel all changes of the LCS orientation. After this button is pressed LCS axes will become codirectional with the respective axes of its own CS.



- ▼ **Read from File.** Allows orienting the LCS on three single vectors, the coordinates of which are written in a file. After clicking this button a dialog will be displayed to specify the file containing coordinates of the vectors, then click the **Open** button. Values in the table will change, and the LCS ghost will turn in the model window. All previously made LCS orientation settings (selection of objects or input of angle values) will be cancelled.

The relation of LCS with the file of single vector coordinates will not be generated.



- ▼ **Save into File.** Allows writing the current coordinates of single vectors in a file. After clicking this button a dialog will be displayed to specify the folder to place the file and its name.

Files of single vector coordinates have text format. You can view, edit or create such file using any text editor. See detailed description of the file of unit vectors coordinates in Section 2.8.3.7.3 on p. 610.

Axes directions



To select an LCS axis to specify its direction, enable the required switch (see Fig.2.8.9):

- ▼ **Guide object for X axis,**
- ▼ **Guide object for Y axis,**
- ▼ **Guide object for Z axis.**

Specify a guide object in the model window or draw a vector that specifies the direction of the selected axis (on vector, refer to Chapter 2.8.5 on p. 613).

The name of the selected object is displayed in the field next to the selected switch. If the vector defines the axis direction, message “Vector” appears in the field.

To the right from the guide object name, there is a button to reverse the axis.

To rotate the LCS around any of its own axes, enable the required radio button in the **Rotation** group, then enter a rotation angle around the selected axis in the **Angle** field.



The **Angle** field is zeroed after every input, and a new LCS rotation angle is counted from its current position.

LCS orientation by the **Axes Directions** method is essentially the same as the **With Respect to CS** method. See description of this process in Section *LCS orientation* on p. 603. You can refer to this section for more information about LCS orientation by defining guide objects and rotation around own axes.

Euler angles system

To define Euler angles defining the position of LCS with respect to its own CS, enter required values in the **Nutation Angle**, **Precession angle** and **Rotation angle** fields (Fig.2.8.10).

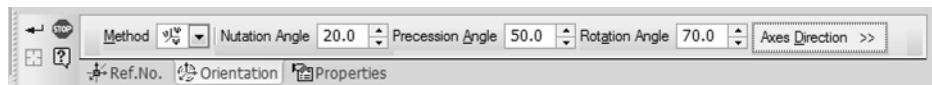


Fig. 2.8.10. Property toolbar for entering Euler angles

When using LCS orientation by the **Euler angles system**, you can reverse any of the axes. For this purpose, having the **<Ctrl>** key pressed, left-click the mouse on the handle at the end of the required axis in the model window. Positions of the LCS ghost will change on the screen, and the values corresponding to the new position will be displayed at angle fields.

By Object

To orient the LCS with respect to some object, the properties of the specified object must provide for unique determination of the LCS orientation.

The list of possible objects is shown in Table 2.8.3 on p. 605. Directions of LCS axes defined by these objects are shown in the *Orientation* column of the same table.

Define the object to orient the LCS in the Design Tree or in the model window. The position of the LCS ghost in the model window changes, the name of the selected object is displayed in the **Object** field. LCS is associated with this object.

2.8.3.7.3. File of unit vectors coordinates

The file of unit vectors coordinates is used to define the LCS orientation. The file has text format (*.txt).

The file of unit vectors coordinates should contain three lines of coordinates: When interpreting the file of unit vectors coordinates it is assumed that:

- ▼ the first line contains coordinates of X vector, the second— Y vector, the third — Z vector;
- ▼ the first number in the line is the vector coordinate along the X axis, the second — along the Y axis, the third — along the Z axis.

All lines located below the lines of coordinates are assumed to be comments and they are not taken into account.



The file can contain coordinates of mutually perpendicular, but non-unit vectors. In this case, LCS is oriented with respect to the specified vectors, and the table shows the values of the respective unit vectors.

Conventions concerning coordinate lines:

- ▼ one line contains coordinates of one vector;
- ▼ delimiters of coordinates in a line are spaces,
- ▼ lines of coordinates are at the beginning of the file and go one after another,
- ▼ at the beginning of the line (before the first coordinate) there can be spaces, at the end of the line (after the last coordinate) — an arbitrary comment.

If the contents of the file indicated for reading vector coordinates fails to meet the above-mentioned requirements, or if the vectors described in it are not mutually perpendicular, a message on the impossibility to retrieve data shall be displayed. The current LCS orientation shall not change in this case.

2.8.3.8. LCS variables

Upon LCS creation, variables of this LCS are automatically generated in the model.



Work with variables and expressions is described in detail in Section 7.1 on p. 1569.

There are twelve informative variables among the LCS variables. They correspond to coordinates of the LCS origin and coordinates of unit vectors. These variables are always created.

Availability and contents of other variables is defined by the LCS creation method:

- ▼ For LCS created by the basic technique, variables of the point of coordinates origin will be generated,
- ▼ For LCS created by additional techniques:

- ▼ if the LCS position was defined by the method of point creation **By Coordinates**, **Transfer**, **On Curve** or **On Surface**, the variables corresponding to the settings specific for these techniques are created for LCS (see Table 2.5.2 on p. 345).
- ▼ If the LCS orientation was defined using Euler angles, the variables corresponding to nutation, precession and rotation angles are created for LCS.



If the LCS's own coordinate system is the local system and not the absolute coordinate system, then only informative variables of the coordinate point of origin and those of unit vectors are created for the LCS.



Values of all LCS's variables are defined in the absolute coordinate system.

Figure 2.8.11 shows variables of LCS built by additional methods, with the LCS position defined by creation of a point **On curve** and orientation — by entering Euler angles.

Name	Expression	Value	Parameter
Part (Bodies-0)			
Origin			
(p)LCS:7			
v30		0.0	Exclude from Calculation
v31		10.0	Start point. X
v32		325.0	Start point. Y
v33		55.0	Start point. Z
v34		10.0	X coordinate
v35		325.0	Y coordinate
v36		55.0	Z Coordinate
X Axis			
v41		0.0	X Axis. X
v42		0.8290	X Axis. Y
v43		0.5592	X Axis. Z
Y Axis			
v45		0.0	Y Axis. X
v46		0.5592	Y Axis. Y
v47		-0.8290	Y Axis. Z
Z Axis			
v49		-1.0	Z Axis. X
v50		0.0	Z Axis. Y
v51		0.0	Z Axis. Z

Fig. 2.8.11. LCS variables

2.8.3.9. LCS deletion

The LCS is deleted in a conventional way using the **Delete** command or the *<Delete>* key (see Section 2.14.3.8 on p. 797).

Please note that removing LCS you also remove the objects that have been built in this LCS. They are derivative objects with respect to LCS being deleted. These objects are listed in the warning dialog (displayed prior to deletion) along with other derivative objects of the LCS being deleted.



If the current LCS is deleted, then the absolute coordinate system shall become the current one.

2.8.3.10. Coordinate system setup

You can make the following settings of a model's coordinate system:

- ▼ for an absolute coordinate system — set names and color of coordinate planes, axes, and origin designations (see Section 9.2.7.5.5 on p. 1860),
- ▼ for local coordinate systems — set names and color of coordinate planes, axes, and origin designations (see Section 9.2.7.5.6 on p. 1860), as well as choose default methods of working with local coordinate systems (see Section 9.1.11.9 on p. 1734).

2.8.4. Control and Connection Points

Control and connection points are special objects used to design pipelines, bundles, etc. Such systems are created using specialized KOMPAS-3D applications (libraries).

Control and connection points are generated in those parts and assembly units that are intended for installation in pipelines, bundles, etc with the means of libraries. Control points provide arrangement of parts and assembly units along the system path, while connection points provide their coupling with each other and with path components. Arrangement and coupling of models is made using couplings.

Control and connection points are created using the **Control Point** and **Connection Point** commands respectively.

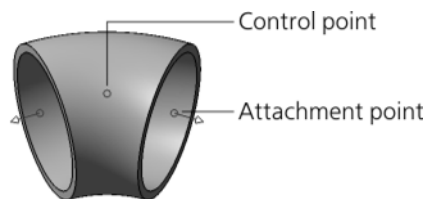


Fig. 2.8.12. Control and connection points



Despite the fact that control and connection points are special objects used by the libraries, you can use them, for example, to mark and designate directions inside models. Control and connection points, as well as the axis (axes) of connection point can be involved in mates just as usual point and linear objects.

2.8.5. Vector



Sometimes you may need to create a vector in space while constructing some object. To do this, the **Vector construction** command is used. In order to call this command, click the **Create Vector** button on the Advanced control panel. The system will switch to creating a vector.

You can draw a vector in space by the following methods:

- ▼ **By two vertices,**
- ▼ **By angle in the CS plane,**
- ▼ **By the CS axis,**
- ▼ **By coefficients,**
- ▼ **By two angles,**
- ▼ **By a straight-line edge, axis or perpendicular to curve plane,**
- ▼ **By a cylinder axis or perpendicular to a flat face, surface,**
- ▼ **Perpendicular to a face at the specified point,**
- ▼ **By basis vector at point of curve,**
- ▼ **Perpendicular to the screen plane.**

Select a vector drawing option from the **Method** list or using commands of the context menu. The elements for managing construction will appear on the Properties Bar (Fig. 2.8.13).

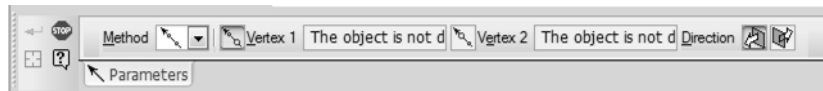


Fig. 2.8.13. Property Bar when you create a vector **By two vertices**.

Set parameters and specify objects used for construction. Parameter input and object specification for each method is detailed in Sections 2.8.5.1— 2.8.5.10.

The vector ghost appears in the model when all the required data have been specified. Phantom length does not depend on the model display scale.



To change objects used for vector construction, press the **Reselect** button on the Special Control Panel and re-select objects.



To finalize the construction, click the **Create Object** button on the Special Control Panel.

The system will resume the object construction process.

2.8.5.1. Drawing by two vertices



By two vertices is a method to draw a vector by two point objects.



By default, the system waits for the vector origin to be specified, with the **Vertex1** switch active. Specify the vector origin in the Model window or in the Design Tree. After the vector origin is specified, the system waits for specification of the point to which the vector will be directed, with the **Vertex2** switch active. Specify the point defining the vector direction in the Model window or Design Tree.



Names of selected point objects will be displayed in the **Vertex1** and **Vertex2** fields.



The ghost vector directed from the first specified point to the second one will be displayed in the screen. This is **Forward Direction**. Activate the **Reverse Direction** switch in the **Direction** group to change the vector direction.



2.8.5.2. Drawing by the angle in the CS plane



Angle in the CS plane is a method to draw a vector by specifying a coordinate system, one of its axes and a vector inclination angle relative to this axis. The vector plane will be defined according to the selected axis:

- ▼ if axis X is specified, the vector angle is measured in XY plane,
- ▼ if axis Y is specified, the vector angle is measured in YZ plane,
- ▼ if axis Z is specified, the vector angle is measured in ZX plane.

The vector origin coincides with the origin of the selected coordinate system.

After you select this drawing method, the ghost vector with the default settings will be displayed in the screen.

Select a coordinate system in the **CS** list to draw a vector.



If the model does not contain LCS, the **CS** list is not available. In this case, the vector will be constructed in the absolute coordinate system.

Select the coordinate system axis to measure the vector angle. To do this, activate the desired switch in the **CS Axis** group:



- ▼ **X Axis**,



- ▼ **Y Axis**,



- ▼ **Z Axis**.

Enter vector inclination angle value in the **Angle** field or select it from the drop-down list.

All updates will be displayed on the ghost vector in the model window.

2.8.5.3. Drawing along CS axis



Along CS axis is a method to draw a vector by selecting a coordinate system and specifying its axis along which the vector will be directed.

After you select this drawing method, the ghost vector with the default settings will be displayed in the screen.

Select a coordinate system in the **CS** list to draw a vector.



If the model does not contain LCS, the **CS** list is not available. In this case, the vector will be constructed in the absolute coordinate system.

The vector origin coincides with the origin of the selected coordinate system.

Select the CS axis to specify the vector direction. To do this, activate the desired switch in the **CS Axis** group:



▼ **X Axis**,



▼ **Y Axis**,



▼ **Z Axis**.



By default, the vector direction coincides with the direction of the selected axis. This is the **Forward Direction**. To reverse the vector direction, activate the **Reverse Direction** switch in the **Direction** group.

All updates will be displayed on the ghost vector in the model window.

2.8.5.4. Drawing by coefficients



By coefficients is a method to draw a vector by selecting a coordinate system and specifying vector expansion coefficients along its coordinate axes X, Y, Z. The expansion coefficients define the vector direction in space.

After you select this drawing method, the ghost vector with the default settings will be displayed in the screen.

Select a coordinate system in the **CS** list to draw a vector.



If the model does not contain LCS, the **CS** list is not available. In this case, the vector will be constructed in the absolute coordinate system.

The vector origin coincides with the origin of the selected coordinate system.

Enter vector expansion coefficients along the coordinate axes of the selected coordinate system in the **X**, **Y**, **Z** fields on the Property Bar.



The resulting direction is the **Forward Direction**. To reverse the vector direction, activate the **Reverse Direction** switch in the **Direction** group.

2.8.5.5. Drawing by two angles



By two angles is a method to draw a vector by selection of a coordinate system and indicating the azimuth and zenith angles that specify vector direction.

After you select this drawing method, the ghost vector with the default settings will be displayed in the screen.

Select a coordinate system in the **CS** list to draw a vector.



If the model does not contain LCS, the **CS** list is not available. In this case, the vector will be constructed in the absolute coordinate system.

The vector origin coincides with the origin of the selected coordinate system.

Vector direction is determined by two angles, A and B. The A angle corresponds to the azimuth angle α of the spherical coordinate system, and the B angle - to the zenith angle β .

The origin of the spherical coordinate system coincides with that of the vector coordinate system.

Azimuth angle α is measured in the XY plane of the vector coordinate system from the X axis counterclockwise (Fig.2.8.14). The OA ray is drawn in the XY plane at the azimuth angle to the X axis.

The OD half-line is drawn in the plane passing through the Z axis and the OC half-line at the zenith angle to the Z axis. This half-line defines the direction of the vector being drawn.

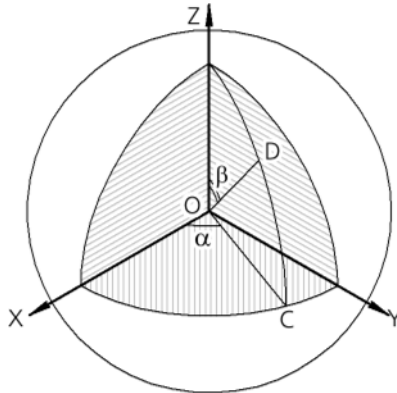


Fig. 2.8.14. Determining vector direction by azimuth and zenith angles

Enter or use the counter to specify azimuth and zenith angle values in the **A** and **B** fields on the Property Bar.

All updates will be displayed on the ghost vector in the model window.

2.8.5.6. Drawing along straight-line edge, axis or perpendicular to curve plane



Along straight-line edge, axis or perpendicular to curve plane is a method to draw a vector by specifying a guide object.

This method can use any rectilinear object as a guide object (see Table 2.1.7 on p. 131) or a flat curve: sketch line, arc, circle, edge in the form of arc/circle/ellipse.

Specify the guide object in the model window or in the Design Tree.

The ghost vector will be displayed.

- ▼ If a straight-line edge or axis is selected as the guide object, the vector will be directed along the selected object.
- ▼ If a plane curve is selected as the guide object, the vector will be directed perpendicular to the curve plane.



You can change direction of the axis being created using the **Direction** group of switches.



2.8.5.7. Drawing along a cylinder axis or perpendicular to a flat face, surface



Along a cylinder axis or perpendicular to a flat face, surface is a method to draw a vector by specifying a guide object.

This method can use any flat object as a guide object (see Table 2.1.7 on p. 131) or any rotation surface, except for the spheric one.

Specify the guide object in the model window or in the Design Tree.

The ghost vector will be displayed.

- ▼ If a flat object is selected as the guide object, the vector will be directed perpendicular to the selected object.
- ▼ If a revolution surface is selected as the guide object, the vector will be directed along the revolution axis.



You can change direction of the vector being created using the group of switches **Direction**.

2.8.5.8. Drawing perpendicular to a face at the specified point



Perpendicular to a face at the specified point is a method to draw a vector perpendicular to the specified face at the set point. Point position is set by the offset of a theoretical surface of the specified face along the U and V isoparametric curves (on the theoretical surface, see Section 2.5.1.3 on p. 328).

By cursor, select the required face in the model.

The following will appear on the selected face: ghost of its theoretic surface and vector ghost. The ghost of the theoretical face surface is displayed as a grid of the U and V isoparametric curves. The vector starts at the face indication point and is perpendicular to it at this point.

The **Face** field of the Property Bar displays the name of the selected face. The input fields **U Parameter, %** and **V Parameter, %** display values of the U and V parameters at the face indication point.

Define the vector origin. For this purpose, enter the required values of the U and V parameters in percent (from 0 to 100) in the **U Parameter, %** and **V Parameter, %** fields or use the counter to specify them. Change in the offset of the vector origin point is displayed on its ghost in the Model window.



By default, the vector direction coincides with the face normal direction at the specified point. This is the **Forward Direction**. To reverse the vector direction, activate the **Reverse Direction** switch in the **Direction** group.

2.8.5.9. Drawing along the basis vector at a curve point



Along the basis vector at a curve point is a method to draw a vector by specifying position of its origin point on a curve and selecting the basis vector of the curve at this point. The selected reference vector defines direction in space of the vector being created.

For more information on basic vectors, see Section 2.5.1.5 on p. 331.

You can select a 3D curve, sketch line, edge or a polyline segment as a curve.

Position of the vector origin is set by displacement along the curve for certain distance from its vertex.

To construct a vector, complete the following steps.

1. Specify the curve to draw a vector in the model window. You can select a 3D curve in the Design Tree.

The **Curve** field of the Property Bar will display the name of the selected object.

Two ghosts will be created on the curve: the tangent vector and the starting vertex of the curve with a directional arrow to count offset. The tangent vector starts at the indication point.

2. Set the vector origin offset along the curve.

The point offset is preset in the range of 0 - 100% of the curve length. The offset in the start vertex is 0.

After the curve is selected, the **% of the curve length** field will display the offset at the indication point. Enter or use the counter to specify the required point offset value.

3. Select vector direction. To do this, activate the desired switch in the **Vector** group:



- ▼ **Tangent Vector**,



- ▼ **Vector of the principal normal**,



- ▼ **Binormal vector**.



The obtained direction is the **Forward Direction**. To reverse the vector direction, activate the **Reverse Direction** switch in the **Direction** group.



2.8.5.10. Drawing perpendicular to the screen plane



Perpendicular to the screen plane is a method to draw a vector perpendicular to the screen plane.



Once this method is selected, the vector ghost will appear in the middle of the screen. This is the **Forward Direction**. To reverse the vector direction, activate the **Reverse Direction** switch in the **Direction** group.



Use the **Fix Position** option to fix the current position of the vector in space.

- ▼ If enabled, this option allows vector to rotate together with the model when you change model orientation.
- ▼ If the option is disabled, vector remains in the middle of the screen being directed perpendicular to it irrespective of any change in the model position.

2.8.6. Composition geometry

Composition geometry is a part of a model which represents a set of objects defining the basic geometrical parameters of the model. For example, it can define the following:

- ▼ assembly spatial domains restricting its components;
- ▼ fastening points;
- ▼ end positions of moving assembly elements (if any),
- ▼ restrictions of dimensions,
- ▼ location of individual parts of an item, etc.

Composition geometry components are, among others, parts, assemblies and local parts.

Parts and assemblies inserted into the model as composition geometry are stored in separate files. In fact, a model contains only links to the files of these parts and assemblies. When migrating such a model to another workplace, the files of all parts and assemblies which compose its composition geometry should also be migrated.

A local part built in the model as a composition geometry is stored directly in the file of this model.

Composition geometry is added into a model from a file of another model (see Section 2.8.6.1.1 on p. 620) or is created in the context of the current model (see Section 2.8.6.1.2 on p. 621).

Then you are able to edit the composition geometry in the context of the current model. When a composition geometry is stored in another file (part or assembly), you can also open it for editing in a separate window. Editing methods are described in Section 2.8.6.2.1 on p. 623.

You can insert a model as a composition geometry into multiple models, and vice versa, a model can contain multiple instances of the same composition geometry.

Composition geometry items (faces, edges, bodies, etc.) can be used when you design various objects in the model. Note that the composition geometry itself does not change.

Since a composition geometry is not a part of an item, the composition geometry is not taken into account when creating reports and specifications for the model.

Composition geometry is not used in determining the MP model.

By default, the composition geometry is placed on a special layer called *Composition geometry*; when there exist multiple layers with this name (you can add or remove these layers), the composition geometry is placed on the smallest one of them. When the layer *Composition geometry* is missing in the model, the system will create it automatically. By default, the automatically created layer is not projected into the associative drawing. You can also enable projecting if required or move a composition geometry to another layer. Working with the dialogs is described in Section 2.15.13 on p. 845.



In general, designing composition geometry is similar to designing components. However, unlike the components, the composition geometry does not support the following operations:

- ▼ enabling/disabling fixing,
- ▼ merging into a subassembly and exploding a subassembly,
- ▼ conversion into a local part using the **Embed in document** command.

The only objects you can merge into a macro item are those included into the components of the composition geometry. You cannot merge these components into a macro item.

2.8.6.1. Adding a composition geometry into a model

You can create a composition geometry either in the current model (“in place”, see Section 2.8.6.1.2 on p. 621) or copy it from another model stored in a file (see Section 2.8.6.1.1).

When working with an assembly, you can use the create/add commands for the composition geometry from the **Operations menu** or by clicking the corresponding buttons on the **Edit assembly** panel. A button to add a composition geometry from a file is also available on the **Edit model** panel.

When working with a part, you can add a composition geometry by clicking the button on the **Edit part** panel. To create a new composition geometry “in place”, enable the **Edit assembly** panel by choosing **View — Toolbars — Edit assembly** or by adding the create command for the composition geometry on any toolbar.

In addition to the methods above, you can also create a composition geometry by converting a component into it (see Section 2.8.6.1.3 on p. 622), as well as by converting model objects into a part (see Section 2.11.2.3.1 on p. 689) or local part (see Section 2.11.2.3.2 on p. 691).

2.8.6.1.1. Adding a composition geometry from a file

To add a part or an assembly from a file into a model as a composition geometry, perform the following.



1. Call the **Operations — Add from file — Component...** command or click the **Add from file** button on the **Model/Detail Editing** or **Assembly Editing** bar.
2. Select the file, containing the required model, in the dialog appeared on screen.

If this part contains variants, geometry and properties of the base part will correspond to one of these variants. The list of all variants, that exist in the model, is contained in the **Variant** list of the file selection dialog. Specify the required variant in the list.

If additional numbers of variants are created in the chosen model, the dialog will also contain the **Additional Number** list. By default, the list has the activated variant under which none of the additional numbers is selected (empty line of the list). If necessary, choose the line with the required number.

3. To finish the file selection, click the **Open** button.

The dialog will close and the current model will be updated with the new composition geometry containing a model from the selected file. The absolute coordinate system of the new subassembly will match the absolute coordinate system of the current assembly.



The absolute coordinate system of the model from the file will match the absolute coordinate system of the current model. To do this, enable the editing mode for this geometry and activate the necessary coordinate grids for the models (for details, see Section 2.8.6.2.2 on p. 623).



In the Design Tree of a model the icon will display in the *Composition geometry* section depending on the type of the composition geometry (part or assembly). The component name, retrieved from it's file, shall appear next to the icon.

The model should be rebuilt after adding a composition geometry (for details on rebuilding, see Section 2.14.3.7 on p. 795).

If the component inserted in the assembly is a model's variant, you can replace it by another variant of the model. For this purpose, call the **Current variant** command from the component's context menu in the Design Tree and specify the required variant in the appearing list of the model's variants. A tick will appear next to the designation of the chosen variant in the list. This will result in inserting the version in the model. Once the version is selected, rebuild the model (for details on rebuilding, see Section 2.14.3.7 on p. 795).

2.8.6.1.2. Creating composition geometry "in place"

To create a new library, do the following.



1. Choose **Operations – Create – Composition geometry...** command or click the **Create composition geometry** button on the **Edit assembly** bar. The **Set up columns** dialog will be displayed (Fig. 2.8.15).

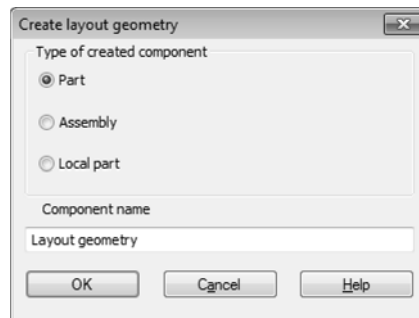


Fig. 2.8.15. The **Create composition geometry** dialog

2. Specify the type of the composition geometry to be created. To do this, select one of the following options in the **Type of component being created** group:
 - ▼ **Part** — to create a part stored in a separate file,
 - ▼ **Assembly** — to create an assembly stored in a separate file,
 - ▼ **Local Part** — to create a local part stored in the current model.
3. Type the name of the composition geometry in the **Component name** field and then click **OK**.



If you are creating a part or an assembly, the file save dialog will appear. Specify the folder and name of the file for the new part or assembly and click **Save**. The Design Tree of the model will display the icon in the *Composition geometry* section denoting the new part or assembly, and the system will switch to the contextual editing of this part or assembly.



If you are creating a local part, you don't need to enter the file name since local parts are stored in the same model. The system immediately switches to the contextual editing of the local part, and in the *Composition geometry* section of the Design Tree the corresponding icon appears.



The absolute coordinate grid of the part, assembly or local part being created is the same as the currently used coordinate grid of the model.

4. Perform all necessary actions to create a part, assembly or local part in the contextual editing mode. Operation in this mode is described in Section 2.11.6.2.1 on p. 724. To exit from this mode, deactivate the **Edit in place** button on the **Current status** panel or choose the **Edit In**



place from the shortcut menu of a part or assembly or click the contextual mode icon in the model window. The system will switch to the current part mode.

Once the contextual editing of a part, assembly or local part is complete, the model should be rebuilt (for details on rebuilding, see Section 2.14.3.7 on p. 795).



For details on how to work with local parts, see Section 2.11.6.5 on p. 726.



If the current model and/or its nested composition geometry (a part, assembly or local part) contains local coordinate grids, then you will be able to relocate the layout model later. To do this, enable the editing mode for this geometry and activate the necessary coordinate grids (for details, see Section 2.8.6.2.2 on p. 623).

2.8.6.1.3. Converting a component into a composition geometry

To convert a model component into a composition geometry, specify the required component in the Design Tree and choose **Editor — Move to composition geometry**. If the Design Tree already contains a section called *Composition geometry*, you can simply drag the component to the icon of this section.

The icon of the selected component in the Design Tree will be moved to the *Composition geometry* section. The component will be assigned the properties of the composition geometry and will become available on the *Composition geometry* layer.

Positions of the component objects in the model window will not change. However, if the absolute coordinate grid of the component matches none of the model's coordinate grids, then the system will create a matching coordinate grid in the model.



If you want to convert multiple components of the model into a composition geometry, select them in the Tree and then choose **Editor — Move to composition geometry**.

You can also perform back conversion of the composition geometry into a component. You can do so when the *Components* section is available in the Design Tree. To perform conversion, drag the composition geometry you want to the icon of this section.

The model should be rebuilt after converting a component into a composition geometry or vice versa (for details on rebuilding, see Section 2.14.3.7 on p. 795).

2.8.6.2. Editing a composition geometry

When editing a composition geometry, you can do the following:

- ▼ edit objects of the composition geometry (see Section 2.8.6.2.1),
- ▼ change parameters of the composition geometry (see Section 2.8.6.2.2 on p. 623),
 - ▼ change the position of the composition geometry in the model,
 - ▼ change the properties of the composition geometry,
 - ▼ choose another source file for the composition geometry,
 - ▼ recalculate the dimensions of the composition geometry with the assigned tolerances.

2.8.6.2.1. Editing composition geometry items

To edit objects of a composition geometry open it in a separate window or switch to the contextual editing mode for the composition geometry. This done, you can now use any editing techniques described in Section 2.14.3 on p. 786.



Note that editing in a separate window is available only when the composition geometry in the model was created from a part or an assembly. You can edit a composition geometry created from a local part only in the context of the current model.



To edit a composition geometry in a separate window, select it in the Design Tree and choose **Edit in window from the shortcut menu**. The source file of the composition geometry will open. You can also open this file as usual by choosing **File — Open**.

Once you make all necessary changes, save and close the source file for the composition geometry or return to the window of the model containing your composition geometry.



You can also save a copy of the composition geometry you edit into another file or location on hard disk drive. Saving methods are described in Section 2.11.6.1.1 on p. 723.



To edit a composition geometry in the context of the current model, select it in the Design Tree and choose **Edit in place** from the shortcut menu. The system will switch into a composition geometry editing mode. Operation in this mode is described in Section 2.11.6.2.1 on p. 724.

To complete editing the component 'in place', use **Edit in place** command once again or release the **Edit in place** button, or click the context editing mode icon in the model window. The system will return to main model operation mode.



Note the following specifics when inserting curve functions.

When a composition geometry is a part or an assembly, the changes are made directly in the file of this part of assembly. That is why the prompt to save the component file appears when editing is complete.

When a composition geometry is a local part, the save file dialog does not display since local parts are stored directly in the model rather than in separate file.

The model should be rebuilt after changing objects of a composition geometry (for details on rebuilding, see Section 2.14.3.7 on p. 795).

2.8.6.2.2. Changing a position and parameters of composition geometry



To change a position of a composition geometry in the model, select it in the Design Tree and choose **Place a component** from the shortcut menu or choose the **Edit** command from the **Editor** menu. Additional controls will become available on the Property bar.



Controls for changing the position are available on the **Location** tab (Fig. 2.8.16).

Fig. 2.8.16. The **Location** tab

On this tab, you can change the position of the composition geometry in the model. The tab provides drop-down lists of coordinate grids of the composition geometry and the current model. These drop-down lists are available when the composition geometry and the current model contain local coordinate grids.

To change a position of a composition geometry, choose the coordinate grids for alignment.

There is a button next to each drop-down list allowing you to specify the necessary coordinate grid from the Design Tree or model window rather than from drop-down lists. You can choose the coordinate grid for the composition geometry by clicking **Component CS**, and choose the coordinate grid for a model by clicking **Current document CS**.

The selected coordinate grids will be overlaid and the position of the composition geometry in the model will change.

You can also configure other parameters of the composition geometry as required.

On the **Properties** tab you can set the color and optical properties of the composition geometry appearance. Control elements used for this are described in Section 2.14.2 on p. 784.

On this tab you can also include or exclude the composition geometry from the calculation. Switches in the **Status** group are used for this.

You can also choose new values of variables in the source model for the composition geometry if the model contains a table of variables. For this, you need to click the **Table of Variables** button and select the required row in the table that appears on the screen.



To configure parameters of the composition geometry without changing its position, you can use the **Component properties** option from the context menu.



To confirm the dimple, click the **Create Object** button on the Advanced Control Panel.

2.8.7. Copies of geometrical objects

A copy of geometrical objects is a model object containing geometrical objects which do not have any history and represent copies of geometrical objects of this model or another one.

The sources of geometrical objects within the same copy must be placed in the same document.

The object in the copy is associated with its source object. All changes in the source object are mirrored in its copy. When the source object is placed in the same document, all changes are mirrored automatically. When the source object and its copy are placed in different documents, you need to rebuild the mode containing a copy in order to mirror the changes (for details on rebuilding, see Section 2.14.3.7 on p. 795).

Each copy of geometrical objects can contain the following model objects:

- ▼ bodies,
- ▼ surfaces,
- ▼ faces,
- ▼ edges,
- ▼ verticals,
- ▼ 3D curves and points,
- ▼ Auxiliary axes and planes,
- ▼ coordinate systems,
- ▼ sketches.

The copy can include objects of the model itself, its components and composition geometry that it contains.

You can change the objects in the copy later as well as use them for building various objects in the model.

2.8.7.1. Creating a Copy of Geometrical Objects



To create a copy of geometrical objects, use the **Copy objects** command from **Operations – Additional controls** or click **Copy objects** on the **Edit model** panel.

If you want to add the objects from the current model into a copy, then you can select the necessary objects either before or after running this command.

If you want to add the objects from another model into a copy, then you can select the necessary objects only after running this command.

The list of possible objects is shown in Table 2.8.7 on p. 624.

When this command has been initiated, **Parameters** and **Properties** tabs appear on the Property bar.

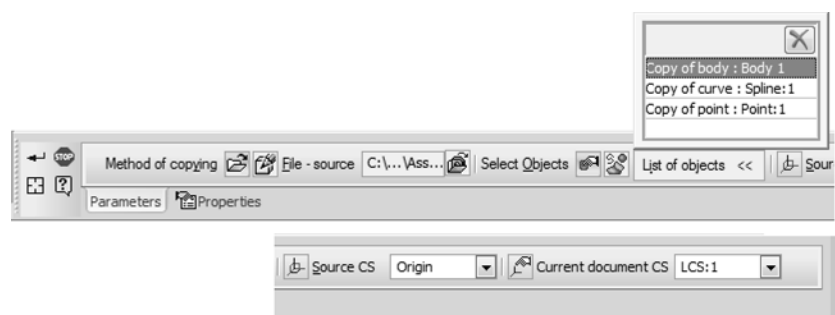


Fig. 2.8.17. **Parameters** Tab

On the **Parameters** tab (Fig. 2.8.17) perform the following actions.

1. Use the **Method** switch group to choose the rectangle's construction method.



By default, the **Context** switch is enabled in the group. In this case, choose the objects for copying in the current model.



To create a copy of objects from another model, activate the **From file** switch. The file selection dialog will appear. Specify the source file for choosing objects and then click **Open**.



The Properties panel will display a field showing the full name of the selected file and the **Source file** button for choosing another file when required. The model window will display the Source model window (see Section 2.8.7.1.1).

- Using switches in the **Object selection** group, specify the method of creating a set of objects included in the copy. There are two options: arbitrary choosing the objects and choosing a set of objects (see Section 2.8.7.4 on p. 629).



By default, the **Manual selection** switch is enabled which corresponds to arbitrary choosing of objects. The names of the selected objects will appear in the **List of Objects** toolbar.

To add an object into a list, specify it in the Design Tree or in the window of the model that you've previously selected as a copy source (current model for the context copying or the model displayed in the window of the source model for copying from file). The selected object is highlighted. Its ghost will appear in the window of the current model.



To remove an object from the list, click it again. You can also delete an object by selecting it on the list and clicking the **Delete from list** button. The **Select other** button empties the list.



You can specify a body or a surface only in the Design Tree.



If another building operation for a body or a surface is specified in the Design Tree, e.g. extrusion, then all faces of this body (surface) will be copied.



To choose a set of objects, activate the **Collections** switch. The Properties bar will display the drop-down list of collections available in the current model (for context copying) or in the model from a file (for copying from file). Select the required measurement unit from the list.



The list of collections is shown on the Properties bar when the document selected for copying contains at least one collection.

Once you've selected objects using one of the above methods, the window of the current model will display the ghosts of the selected objects.

- Specify the position of the copy in the model. For this, do the following.



▼ When the objects are copied from the current model, the copy is placed in the current coordinate grid of the model by default. To change the position of the copy, make another coordinate grid the default one. To do this, select it from the **Current document CS** list or click the corresponding button and specify the coordinate grid you want in the Design Tree or a model window.



▼ When the objects are copied from another model, the absolute coordinate grid of the source model by default matches the current coordinate grid of the model where the copy is being created. To change the position of the copy, specify another coordinate grid for overlaying. To do this, choose them from the **Source CS** and **Current document CS** lists. You can also use the buttons on the Properties bar to choose the coordinate grids. Once you click the button, you will be able to specify the option in the Design Tree

or window of the current model (for the coordinate grid of the current document) and in the Design Tree or in the window of the source model (for the coordinate grid of the source model).



The controls for choosing the coordinate grids are available when the current model and/or source model contain local coordinate grids.

4. On the **Properties** tab, specify the name of the new copy of objects.



5. To finish the construction of the curve click the **Create Object** button on the Special Control Panel.



In the *Copies* section of the Design Tree of the model, the icon and name of the created copy of object will appear, and in the window of the current model, the objects included in this copy will display.

2.8.7.1.1. Variables Window

The window of the source model is displayed in the window of the current model when another model file is used for a source of the objects included in the copy.

This window contains an image of the source model and the Design Tree of the current model (Fig. 2.8.18).

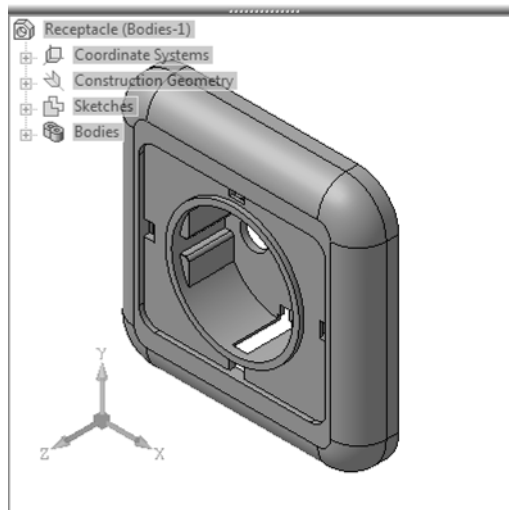


Fig. 2.8.18. Window of the Source Model

The source model's Design Tree displays the following objects:

- ▼ coordinate systems,
- ▼ construction geometry,
- ▼ layout elements (unavailable for copying),
- ▼ curves and points,
- ▼ sketches,

- ▼ bodies constructed directly in the source model,
- ▼ 'dimension' objects (unavailable for copying).

You can specify objects in the window of the source model as usual by clicking them in the Tree or directly in the window.

You can move the source model window by dragging it by the header. You can also resize the window by clicking and dragging its borders. Moving or resizing the window are only possible within the window of the current model.



Type of displaying objects in the window (wireframe, shaded, etc.) corresponds to the selected in the current model window.



You can also zoom in or zoom out the objects in the window using the **Show All** and **Current Zoom** commands, as well as by scrolling the mouse wheel.

You can also control the display of the objects in the window using the commands **View — Hide**.

You can also rotate the model in the window using your keyboard (see *Rotating a model using the keyboard* on p. 120) or mouse (see *Rotating the model using the mouse* on p. 120).

2.8.7.2. Editing a Copy of Geometrical Objects



To edit a copy of geometrical objects, select it in the Design Tree and choose **Edit** from the **Editor** menu or from the shortcut menu. The same controls as those available for inserting a component to the assembly will be active on the Property bar.

The same controls as those available for creating a copy will be active on the Properties bar. This can be done using the same method as for copy creation (see Section 2.8.7.1 on p. 625).

2.8.7.3. Explosion of a Copy of Geometrical Objects

A copy of geometrical objects contain objects associated with the source objects, but without history. You can explode an existing copy into separate objects. After explosion, the links between the objects in the copy and the source objects will be disposed. The model will contain a set of objects without history.

To explode a copy, select it in the Design Tree and then choose **Explode** from the **Editor** menu or from the shortcut menu.



To explode multiple copies, select them in the Design Tree and then choose the command from the **Editor** menu.

Upon calling the command, an explode dialog will appear. To confirm the breaking, press the **OK** button. The **Cancel** button closes the dialog and rejects all changes.

After explosion, the icon of the exploded copy will be removed from the model. The copies of the objects are displayed in the Design Tree separately and preserve their position within the model.

2.8.7.4. Collections

A collection is an arbitrary set of objects. Usually, collections are used for storing objects required for certain building operations. For example, a collection can contain faces of the adjacent surfaces and/or other faces of the surrounding parts which define the position and configuration of the newly created component.

Collections are used for picking objects when creating copies of geometrical objects (see Section 2.8.7 on p. 624). They allow you to quickly pick the necessary objects without clicking them in the Design Tree or model window.

A collection can contain the following model objects:

- ▼ bodies,
- ▼ surfaces,
- ▼ faces,
- ▼ edges,
- ▼ verticals,
- ▼ 3D curves and points,
- ▼ Auxiliary axes and planes,
- ▼ coordinate systems,
- ▼ sketches.

A collection can also include the objects from the current model, its components or its nested composition geometry.



To create a collection, choose **Operations — Additional Controls — Collection** or click the **Collection** button on the **Edit Model** panel.

Parameters and **Properties** tabs will appear on the Property bar.

The **Parameters** tab provides the **List of Objects** panel allowing you to create a set of objects to be included into the collection. You can select the surfaces either before or after calling the command.

To add an object to the list select it in the Design Tree or in the Model window.



You can specify a body or a surface only in the Design Tree.



To remove an object from the list, click it again. You can also delete an object by selecting it on the list and clicking the **Delete from list** button. The **Select other** button empties the list.



On the **Properties** tab, specify the name of the new collection.



To finish the construction of the curve click the **Create Object** button on the Special Control Panel.



The Design Tree will display the icon of the new collection and its name. When you select a collection in the Design Tree, all its objects are highlighted in the model window.



When creating a collection in the model, no copies of the selected objects are created. A new collection will contain the original objects.



You can change the set of objects in the collection and its name later. To do this, select a collection in the Design Tree and choose **Edit** from the **Editor** menu or from the shortcut menu. The same controls as those available for inserting a component to the assembly will be active on the Property bar. The collection can be modified in the same way as it is created.

2.9. Dimensions, designations, thread (notation)

2.9.1. General Information

KOMPAS-3D allows creating different sets of dimensions and designations in 3D models.

Commands for creating dimensions and symbols are grouped in **Operations— Dimensions and Process Symbols** menu, and the respective buttons for these commands can be found on the **Dimensions and Process Symbols** toolbar (Fig. 2.9.1).



Fig. 2.9.1. **Dimensions and Process Symbols** toolbar

Created dimensions and designations can be transferred to a drawing when creating a model's associative views in it (see Section 3.6.1.2 on p. 1208).

Dimensions and designations transferred to the drawing are **projection designations**. Work with projection designations is described in Section 3.6.3.5 on p. 1226.

2.9.2. Dimensions

2.9.2.1. General information on dimensions

Dimensions in a model can be of two types: controlling and informational. The dimension's type defines whether the dimension will affect the model's object or, vice versa, the object will affect the dimension.

The **controlling dimension** is a dimension that controls the geometry of the model's object. The value of the controlling dimension can be changed by the user. After that, the object with the dimension is rebuilt to satisfy this new dimension value. The rectangular frame around the value indicates the controlling dimension. This frame is not printed out.

The **informational dimension** is a dimension controlled by the model's object. It is impossible to change the value of the informational dimension. If the object, to which the informational dimension is assigned, has been rebuilt, the dimension will be rebuilt with recalculation of its value.

When working with a model, you can use **dimensions of sketches and operations**. Besides that, you can create **dimensions that are layout elements**.

- ▼ **Dimensions of sketches and operations:** they are not displayed in the Design Tree; they are visible only during creation of an operation or in the special mode; they can be both controlling and informational; they are not transferred to the drawing; more information about dimensions of sketches and operations can be found in Section 2.9.2.2 on p. 632.
- ▼ **Dimensions — layout elements:**

- ▼ **derivative dimensions** are dimensions of sketches and operations represented as layout elements; they inherit the type of respective dimensions; in the Design Tree, they are displayed as subordinate objects of sketches and operations; they can be transferred to the drawing; more information about derivative dimensions can be found in Section 2.9.2.8 on p. 648;
- ▼ **dimensions manually assigned to a model**; they are always informational; in the Design Tree, they are displayed as individual objects of the model; they can be transferred to the drawing; more information about manually assigned dimensions can be found in Section 2.9.2.3 on p. 637.



Appearance of derivative dimensions and manually assigned dimensions are the same (for example, arrow length and angle, font and position of the dimension text and so on), since it is controlled by the model dimensions settings (see Section 9.2.7.2 on p. 1836). Display color is the only difference between derivative dimensions and manually assigned dimensions (by default, the derivative dimensions color is violet, while the color of manually assigned dimensions is brown). See Section 9.2.7.4.7 on p. 1850 for more information about adjusting the color of model objects' display.

2.9.2.2. Dimensions of sketches and operations

In KOMPAS-3D, most operations have a certain set of numeric parameters. If an operation parameter's value is expressed in linear or angular values, the parameter can be displayed as the dimension of the respective type – an **operation's dimension**. For example, numeric parameters of the extrusion operation are as follows: extrusion distance, slope angle, and wall thickness.

Dimensions of operation are displayed on the screen in the following cases:

- ▼ when performing an operation;
- ▼ in the special **mode of displaying the selected element's dimensions** (see Section 2.9.2.2.2 on p. 635).

For each numeric parameter of the operation, a variable is created automatically. The variable's name is displayed in parenthesis below the dimension line of the dimension that corresponds to the operation's parameter (Fig. 2.9.2).

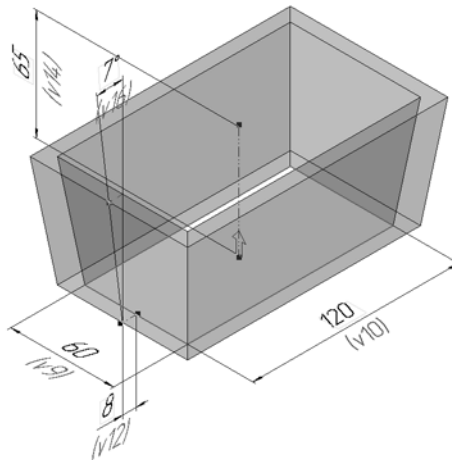


Fig. 2.9.2. Example of extrusion element construction

Dimensions of a sketch with variables can also be viewed in the dimensions display mode. Besides that, sketch dimensions are displayed when performing the operation if the sketch is involved in its creation.



When performing an operation, dimensions of sketches and operations are displayed on the screen, if the dialog for customization of display parameters of dimensions and designations (see Section 9.1.11.12 on p. 1736) has the **Dimensions of sketches and operations** option enabled.



When working with an array, you can enable the display of the dimensions of sketches and operations for each copy of the array (see Section 2.7.2.4 on p. 536).

Dimensions of sketches and operations can be both controlling and informational (see Section 2.9.2.1 on p. 631).

Dimensions of sketches and operations are not displayed in the Design Tree.

You can enable or disable displaying the names of variables and the “controlling” sign in the dimensions of sketches and operations (see Section 2.9.2.2.3 on p. 637).

If necessary, you can change the value of the dimension of the sketch or operation. Changing the dimension’s value is possible both when performing the operation (see Section 2.1.4.2.3 on p. 139) and after it in the mode of displaying the dimension chosen for elements (see Section 2.9.2.2.1 on p. 634).

If necessary, you can display dimensions of sketches and operations as layout elements (see Section 2.9.2.8 on p. 648). As a result, **derivative dimensions** corresponding to dimensions of sketches and operations will appear in the Model Window. In the Design Tree, derivative dimensions are displayed as subordinate to sketches and operations (see Fig. 2.9.18 on p. 648).

2.9.2.2.1. Setting the dimension's value

You can edit the values of the following dimensions:

- ▼ associative dimensions of a sketch and dimensions of an operation when performing the operation or in the mode of displaying the selected element's dimensions (for more information about the mode, see Section 2.9.2.2.2 on p. 635);
- ▼ manually assigned dimensions (see Section 2.9.2.3 on p. 637),
- ▼ dimensions of a sketch or operation presented as layout elements (derivative dimensions; see Section 2.9.2.8 on p. 648).



When performing an operation, dimensions of a sketch and operation are displayed on the screen, if the dialog for customization of display parameters of dimensions and designations (see Section 9.1.11.12 on p. 1736) has the **Dimensions of sketches and operations** option enabled.

Enter a dimension value in the dimension value setting dialog (Fig. 2.9.3).



Fig. 2.9.3. Dialog to set dimension value: a) operation, b) sketch

The dialog appears on the screen after:

- ▼ double-clicking the dimension's dimension text;
- ▼ calling the **Set dimension value** command from the context menu of the derivative dimension selected on the Design Tree or in the model window.

In this dialog you can perform the following actions.

- ▼ Edit the name of a variable (only for sketch dimension and manually-assigned dimension).
- ▼ Input desired dimension value¹: number, constant, or expression for calculation of the dimension value (the expression syntax is described in Section 7.1.3.3.1 on p. 1580).



- ▼ Assign individual tolerance using the **Assign tolerance** button:
 - ▼ for the value of an operation's parameter,
 - ▼ for a sketch dimension,
 - ▼ for a manually-set dimension.

Assignment of tolerances is described in Section 2.10.2.2 on p. 669.

- ▼ Input a comment for a variable.



- ▼ convert the controlling dimension into informational or vice versa, using the button **Info Dimension** (only for sketch dimension),



- ▼ enable automatic rebuilding of a model after setting the dimension's value (closing the dialog) using the **Rebuild** button.

1. For the sketch's dimension, you can enter the value only if the dimension is the controlling one. Otherwise, the field for the dimension's value entry will not be available.



Upon completing dimension editing, press the **Create dimension** in the dialog.



- ▼ If automatic model rebuilding is enabled in the dialog, the image on the screen will be rebuilt in accordance with the new value of the dimension.
- ▼ If automatic model rebuilding is disabled, the image on the screen will no be changed. In order to rebuild the model, click the **Rebuild** button on the **View** bar or press <F5>. The image on the screen will be rebuilt in accordance with the new value of the dimension.



Note the following peculiarity when editing the value of the dimension of a sketch or operation *in the process of performing the operation*. Once the dimension value in the **Set the dimension's value** dialog is changed, the ghost of the object being created will be rebuilt irrespective of the **Rebuild** button status (see above).

2.9.2.2.2. Displaying dimensions of sketches and operations in the Model Window

When working with the model, you can view:

- ▼ controlling dimensions of the operation, that creates the object, without entering the process of its editing,
- ▼ controlling dimensions of a sketch with variables outside the mode of the sketch editing.

For this purpose, the **special mode of displaying dimensions of the selected element** is used.

Besides that, you can change the values of dimensions of sketches and operations in this mode.



In order to enable the mode of displaying dimensions of the selected element, click the **Dimensions of selected element** button on the **Modes** bar or select its name in the **View** menu. The **Dimensions of the Selected Element** button is available if none of the 3D construction commands is active.

Choose the required object by selecting it in the Design Tree or specify any of its primitives in the Model Window: a face, edge, or vertex.

Dimensions of the highlighted object will be displayed on the screen (Fig. 2.9.4).

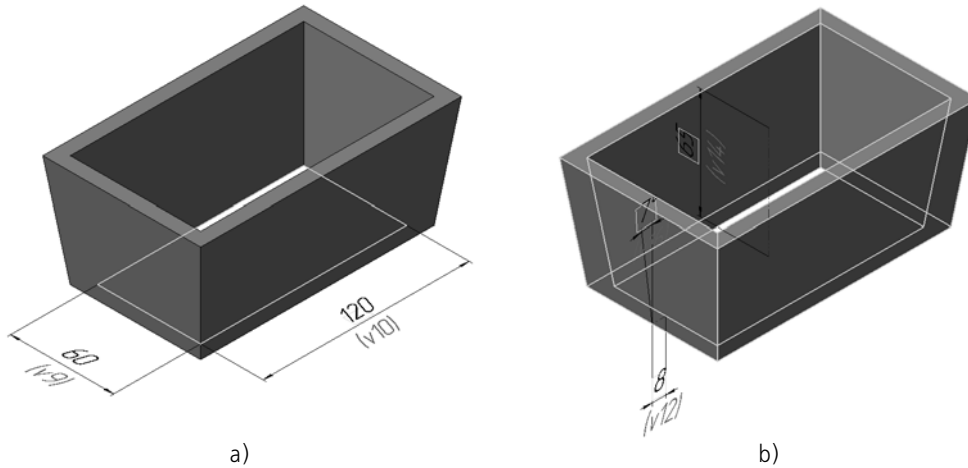


Fig. 2.9.4. Example of displaying dimensions of a selected object:
a) sketch, b) extrusion element



If an object is selected, and its dimensions are not displayed in the Model Window, it means that:

- ▼ the operation has no numeric parameters expressed in linear or angular values;
- ▼ the sketch has no controlling dimensions, or the sketch's controlling dimensions do not have a variable.

To adjust the dimension's value, double-click its dimension text. The dialog to set the dimension value will be displayed. Specify the required dimension value in this dialog (see Section 2.9.2.2.1 on p. 634).

Peculiarities of displaying dimensions of a selected object

1. Viewing and editing are only available for the dimensions of those objects that were built directly in the model (not in any of its components). To work with a component, you need to go into the mode of in-window editing or that on the spot.
2. If an object's dimension is already presented as a derivative dimension (see Fig. 2.9.18 on p. 648), the image of the dimension (a copy of the derivative one) will not be formed in the mode of displaying dimensions.
3. Connecting lines of dimensions (for example, lines that conditionally indicate objects with dimensions, see Fig. 2.9.4, b) are displayed on the screen if the dialog for dimension and designation display setup (see Section 9.1.11.12 on p. 1736) has the **Connecting lines** option enabled.

2.9.2.2.3. Displaying names of variables in dimensions of sketches and operations

By default, the mode of displaying names of variables and the “controlling” attribute in the dimensions of sketches and operations is enabled in the model (more information about controlling dimensions can be found in Section 2.9.2.1 on p. 631).

In this mode, dimensions of sketches and operations are displayed as follows:

- ▼ values of dimensions are displayed in frames (see Fig. 2.9.4 on p. 636),
- ▼ names of variables corresponding to dimensions are displayed under their dimension lines.

If the value of an operation's numeric parameter or sketch dimensions is defined by a constant, expression, or negative number, the following information is displayed under the dimensional line of dimensions: name of the variable and its value.



In order to disabled the mode of displaying names of variables in dimensions of sketches and operations, call the **View — Names of variables in dimensions** command or press <F11>.

2.9.2.2.4. Specifics of Working with Dimensions of Operations

When editing an operation, an of its numeric parameters can become dependent on the model's object. In this case, the operation's dimension, that represents this parameter, becomes the informational one and will no longer be displayed on the screen. If this operation's dimension was presented as a layout element (derivative dimension), the derivative dimension will be deleted once editing of the operation is finished.

For example, when editing an extrusion operation, the **To Distance** construction method was replaced by the **To Vertex** method. The extrusion distance will become dependent on the position of the specified vertex. The size of operation, corresponding to the extrusion distance, becomes an informational one and will no longer be displayed on the screen. The derivative dimension, representing this operation's dimension, will be deleted.

2.9.2.3. Procedure for manual assignment of dimensions

The system allows manually assigning linear, angular, radial and diameter dimensions to objects of various types. The object to which the dimension is attached is called base object.

All dimensions manually-assigned in 3D parts and assemblies are informational dimensions (see Section 2.9.2.1 on p. 631).

To create a dimension use the appropriate dimensioning command.



If automatic dimensioning is required, click **Object Autocreation** button on the Advanced Control Panel. The automatic and manual object creation is detailed in Section 1.4.2.10 on p. 93.

If necessary, you can make more than one dimension of the same type in one dimensioning command. To do this, without leaving the command, select the objects one by one which the dimensions should be attached to. The objects can be selected in the Design Tree or in the Model window.

The created dimension will appear in the model window, while their icons will appear in the Design Tree.



To finish current command, click **Interrupt Command** button on the Advanced Control Panel.



Select Other button makes it possible to reselect objects.

Controls located in the Properties tab allow defining name of dimension in the Design Tree and color of drawing of dimensional lines.



When dimensioning with default settings, the dimension text may appear inside the body. To facilitate further manipulations with the dimension, use mouse to move the handle that determines the dimension text position so that the dimension text is displayed outside of the part.



Once attached, each dimension is automatically assigned a variable name. To change the default variable name, double-click the desired dimension's text. A **Set Value of Dimension** dialog will appear. Enter the new variable name and click **Create Dimension** button. More information about the dialog for dimension value setting can be found in Section 2.9.2.2.1 on p. 634.



The 3D model operation mode doesn't support vector fonts including the KOMPAS-3D supplied fonts GOST type A (plotter), GOST type B (plotter), Symbol type A (plotter), Symbol type B (plotter) therefore you shouldn't use them for dimension text.

Dimensions can sometimes disturb from viewing the model. To hide a dimension, select it and use **Hide** context menu item. A dimension will also become hidden if any of its objects are hidden.

Hidden dimensions will not be transferred to the drawing when associative views of the model are created in it.

To make a hidden dimension visible, select it in the Design Tree and use **Show** context menu item.

KOMPAS-3D uses such default symbol settings that are most commonly used in design documentation. If necessary, you can change these values. Dimension parameter customization for model is described in Section 9.2.7.2 on p. 1836.

2.9.2.4. Linear Dimension

For linear dimensioning, point, line and planar objects can be used.

Point objects are:

- ▼ sketch point,
- ▼ 3D point,
- ▼ 3D line vertex,
- ▼ Solid or surface vertex.

Straight objects:

- ▼ line in a sketch,
- ▼ polyline segment,
- ▼ solid or surface edge.

Flat (planar) objects

- ▼ coordinate plane,

- ▼ auxiliary (construction) plane,
- ▼ Face of the body or surface.



To create linear dimensions use the **Linear Dimension** command.

Select with the cursor an objects to set dimension.

- ▼ To attach a dimension to a straight object, select the straight object with the mouse.
- ▼ To insert a dimension between two point objects, two planar objects or a point object and a planar object, select the objects with the mouse.



Linear dimensioning between two planar objects is only possible if the objects are parallel to each other.

Select the base plane—the plane which the dimensioning plane will be parallel to.

When attaching a dimension to an edge or a line the base plane is selected by the system automatically. In the first case one of the pairs of faces divided by an edge is used as the base plane, in the second case — sketch plane.



You can change the base plane (including the automatically selected one) without exiting the command. For this enable the Plane switch on the Properties toolbar and define the required flat object.

After defining objects and base plane a ghost of the dimension will be displayed. The dimensioned segment is displayed as a dash-dot line with two points. Its ends are the dimension snapping points (points of extension line origin). The projection line of one of objects on the dimension attaching plane is displayed as a dash-dot line (the necessity of projection is defined by mutual position of objects and dimension attaching plane).



The dimensioned segment and the projection relation line will be displayed if **Show Connection Lines** option is turned on in dimension display settings (see Section 9.1.11.12 on p. 1736).

Set the position for the dimension line. For this enter the value of leader length in **Length** field of **Dimension** Tab on Properties panel (the value may be both positive and negative) or use the mouse to define points on its ends.

If necessary, edit the dimension text (see Section 3.3.1.3 on p. 979) and specify the dimension display parameters (see Section 3.3.1.2 on p. 977) using Properties bar tabs.

If the dimension text should be placed on a leader shelf, follow the procedure described in Section 2.9.2.4.1.



By default, linear dimension will be placed in the dimensioning plane to the left of the dimensioned segment. The left side of the dimensioned segment is determined as follows: A vector from the first dimension snap point to the second snap point (or its projection) is constructed. The half of the dimensioning plane to the left of this vector is considered the left side.



When adding more than one dimension in a single dimensioning command, the selected base plane will be used for all of them. To select another base plane, click **Select Other** button on the Property bar.

An example of linear dimensioning between two point objects is illustrated in Fig. 2.9.5.

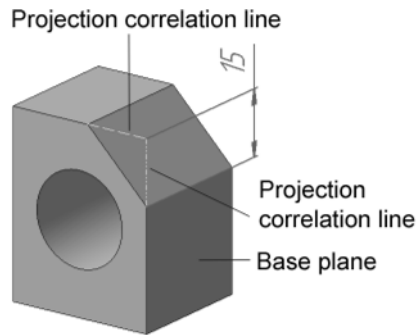


Fig. 2.9.5. Linear dimensioning example

2.9.2.4.1. Placing Dimension Text on Leader Shelf

If necessary, dimension text can be displayed on a leader shelf. This can be achieved by doing the following when creating or editing a dimension.

1. Activate page MP **Parameters** of the Property Bar.
2. In the **Text Location** drop-down list, select one of the following: **On Leader Shelf Left**, **On Leader Shelf Right**, **On Leader Shelf Top** or **On Leader Shelf Bottom**.

The leader line will appear (Fig. 2.9.6).

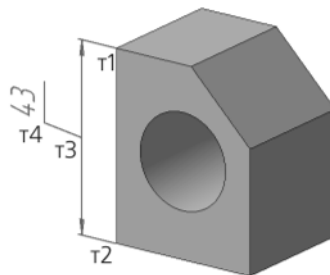


Fig. 2.9.6. Linear dimension on leader shelf

3. Set the leader line start position and leader shelf start by moving **p3** and **p4** points with your mouse.

2.9.2.4.2. Dependency of the dimension value on the base plane selection

In all cases of linear dimension attaching, in addition to the dimension between flat and point objects, the base plane defines not only space position of dimension, but also its value.

Let's examine attaching dimensions between point objects in the model shown in Fig. 2.9.7.

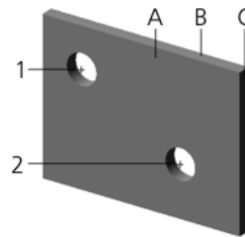


Fig. 2.9.7. Point objects and possible base faces to attach dimensions

Points 1 and 2, which belong to A edge and are located in the centers of round edges are selected for dimension creation.

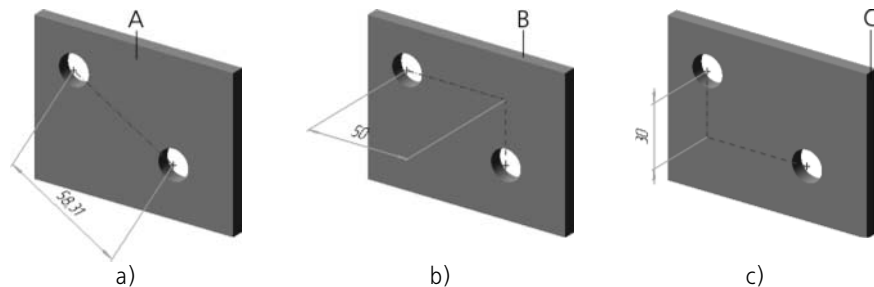


Fig. 2.9.8. Dependency of the position and dimension value on the base plane selection:
a) base plane— A face; b) base plane— B face c) base plane— C face

- ▼ The figure 2.9.8, a) shows the dimension which base plane is A face. Since this face contains both points 1 and 2, it also is the plane to attach a dimension. The dimension value is equal to the actual distance between points 1 and 2.
- ▼ Figure 2.9.8 b) shows the dimension the base plane of which is B face. The plane to attach the dimension is parallel to this face and passes through point 1. Point 2 is projected on the dimension attaching plane. The dimension value is equal to the distance from point 1 to the projection of point 2, i.e. the length of projection of the actual distance between points 1 and 2 to the base plane.
- ▼ Figure 2.9.8 c) shows an example, the base plane of which is C face. Its construction is similar to the previous one.

2.9.2.5. Linear Dimension from Line to Point

Linear dimension from a line to a point is created between a straight object and a point. Following objects can be used.

Straight objects:

- ▼ line in a sketch,
- ▼ polyline segment,
- ▼ coordinate axis,
- ▼ auxiliary axis,

- ▼ solid or surface edge.
On-Spot:
- ▼ sketch point,
- ▼ 3D point,
- ▼ Origin
- ▼ 3D line vertex,
- ▼ Solid or surface vertex.



To create linear dimensions from a line segment to a point use the **Linear Dimension from Line to Point** command.

Select with the cursor a straight object and then a point.

The dimension will appear in the plane running through the objects (Fig.2.9.9).

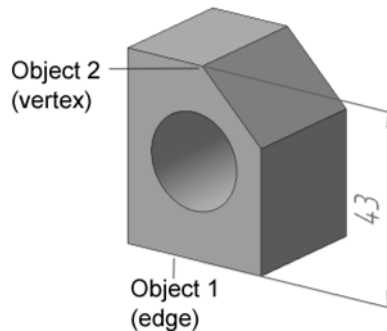


Fig. 2.9.9. Example of linear dimensioning from segment to point

Set the position for the dimension line. For this enter the value of leader length in **Length** field of **Dimension** Tab on Properties panel (the value may be both positive and negative) or use the mouse to define points on its ends.

If necessary, edit the dimension text (see Section 3.3.1.3 on p. 979) and specify the dimension display parameters (see Section 3.3.1.2 on p. 977) using Properties bar tabs.

If the dimension text should be placed on a leader shelf, follow the procedure described in Section 2.9.2.4.1.

2.9.2.6. Angular Dimension

For angular dimensioning, straight and planar objects that are sides of an angle are used.

Straight objects:

- ▼ line in a sketch,
- ▼ polyline segment,
- ▼ auxiliary axis,
- ▼ solid or surface edge.

Flat (planar) objects

- ▼ coordinate plane,

- ▼ auxiliary (construction) plane,
- ▼ Face of the body or surface.



To create angular dimensions use **Angular Dimension** command.

Select an objects with a cursor. two straight objects, two planar objects, or one straight and one planar object.

When creating dimensions between two straight objects, the dimension will be created in the plane running through these objects (Fig. 2.9.10,a). If the selected objects do not belong to the same plane, no dimension will be created.

When creating dimensions between two planar objects, the dimension will be created in the plane perpendicular to the intersection of these objects (Fig. 2.9.10,b). Dimensioning plane projections on the selected planar objects will be displayed as dashed lines.



The dimension between two planar objects can be moved along the intersection of these objects. To do this move the dimension's vertex handle with the mouse as necessary during dimension creation or editing.

When creating dimensions from a straight object to a planar object, the dimension will be created in the plane running through the selected straight object and its projection on the selected planar object (Fig. 2.9.10,c).



The projection lines will be displayed if **Show Connection Lines** option is turned on in dimension display settings (see Section 9.1.11.12 on p. 1736).

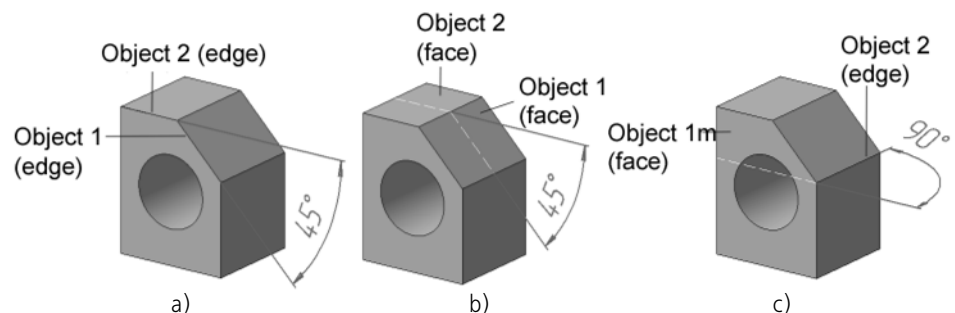


Fig. 2.9.10. Examples of the angle dimension:
a) between two edges, b) between two faces, c) between an edge and a face



The type of the newly created angular dimension will be determined by the system automatically. If necessary, you can change the dimensioning method chosen by the system using **Type** option group on the **Dimension** tab of the Property bar, including enabling dimensioning of angles greater than 180° (automatic choice of this option is disabled).

Set the position for the dimension line. To can do this by entering the extension line length in **Length** field (the length can be either positive or negative) or by resizing the handles at its ends using your mouse.

To mirror the display of dimension lines while keeping their lengths, click **Change Orientation** button.

If the dimension text should be placed on a leader shelf, follow the procedure described in Section 2.9.2.4.1 on p. 640.

If necessary, edit the dimension text (see Section 3.3.1.3 on p. 979) and specify the dimension display parameters (see Section 3.3.1.2 on p. 977) using Properties bar tabs.

2.9.2.7. Radial and Diameter Dimensions

The system allows for radial and diameter dimensioning in a number of ways. The settings for these two dimensioning types are mostly the same. The difference is in dimension type selection (see Sections 2.9.2.7.2 and 2.9.2.7.3).

2.9.2.7.1. Selection of objects for dimension creation

Radial and diameter dimensioning uses following object types:

- ▼ Circle (circular arc) in sketch
- ▼ Circular (arcuate) solid edge or surface edge
- ▼ Circular, conical, spherical or toroidal solid face or surface face.

When selecting a circle in a sketch, dimension is created in the sketch plane.

When selecting an edge, dimension is created in the plane of this edge (Fig. 2.9.11, a, b).

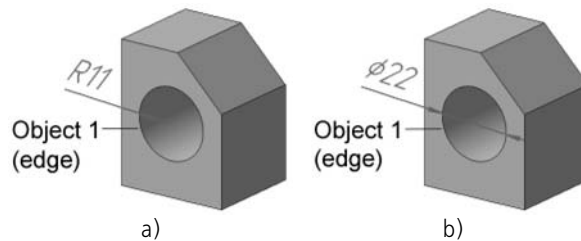


Fig. 2.9.11. Examples of part edge dimensioning:
a) radial, b) diameter

When selecting a face, dimension is created in the plane perpendicular to an axis of selected face (Fig. 2.9.12, a, b).

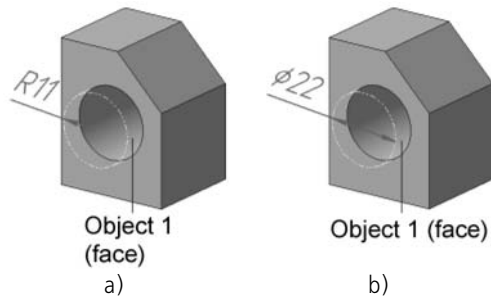


Fig. 2.9.12. Examples of part face dimensioning:
a) radial, b) diameter

The dimensioned circle will be displayed as a dash-dotted line with two points. The extension lines will be displayed as dashed lines.



The dimensioned circle and the extension lines will be displayed if **Show Connection Lines** option is turned on in dimension display settings (see Section 9.1.11.12 on p. 1736).

Dimension position can be free or fixed.

For free choice of dimension position, move the handle in the center of the dimensioned circle with your mouse along the face axis.

To fix the dimension position, turn on **Position** option on the **Dimension** tab of the Property bar and select the object to fix to: a point object or a planar object parallel to the dimensioning plane. The dimension will be created in the plane running through the fix object. Selecting this object again will cancel dimension position fixing.

An associative relation will be established between the dimension and the fix object. This relation causes the dimension to follow the fix object wherever it is moved.

2.9.2.7.2. Radial Dimension



To create radial dimensions use the **Radial Dimension** command.

Select an objects to set dimension (see Section 2.9.2.7.1).

The radial dimension can be created either from the circle center or not from the circle center. In both cases, the dimension line belongs to a line going through the center of the circle being dimensioned. The difference is the following. If a dimension is created from a center, then its dimension line length cannot be less than the radius. If a dimension is created not from a center, then its dimension line can be of an arbitrary length.



To select the desired dimensioning option use the **Type** option group on the **Dimension** tab of the Property bar.

Specify the dimension line position by moving the handle **p1** with your mouse (Fig. 2.9.13).

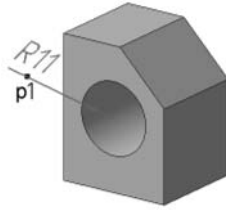


Fig. 2.9.13. Radial dimensioning with manually positioned dimension line

If necessary, edit the dimension text (see Section 3.3.1.3 on p. 979) and specify the dimension display parameters (see Section 3.3.1.2 on p. 977) using Properties bar tabs.

If the dimension text should be placed on a leader shelf, follow the procedure described in Section 2.9.2.4.1 on p. 640.

2.9.2.7.3. Diameter Dimension



To create diameter dimensions use the **Diameter Dimension** command.



Select an object to set dimension (see Section 2.9.2.7.1).



A dimension line can be Continuous or Broken. The desired option can be selected in the **Type** option group on the **Dimension** tab of the Property bar.



A broken dimension line projects (extends) beyond the circle center by the distance equal to 1/5 of the radius, but not less than by the distance set in this document for the dimension line projection (extension) beyond the text. This value is specified during setup dimension parameters in the current model (see 9.2.7.2.2 on p. 1836).

Specify the dimension line position by moving the handle **p1** with your mouse (Fig. 2.9.14).

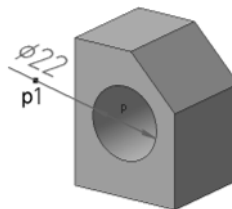


Fig. 2.9.14. Diameter dimensioning with manually positioned dimension line

If necessary, edit the dimension text (see Section 3.3.1.3 on p. 979) and specify the dimension display parameters (see Section 3.3.1.2 on p. 977) using Properties bar tabs.

If the dimension text should be placed on a leader shelf, follow the procedure described in Section 2.9.2.4.1 on p. 640.

2.9.2.7.4. Dimensioning Features

Dimensioning of a conical, spherical or toroidal face has the following features.

Conical face

The conical face diameter dimension is created in the plane perpendicular to the face axis (cone axis) and running through the point where the face was selected (Fig.2.9.15).

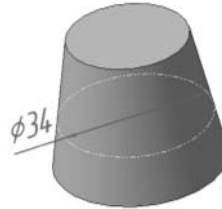


Fig. 2.9.15. Conical face diameter dimensioning example

The dimension will correspond to the radius (diameter) of the cone's section by the dimensioning plane. When the dimension is moved along the cone axis, it will update as the cone section radius (diameter) changes.

If the dimension position is fixed (see Section 2.9.2.7.1 on p. 644), then the dimension will correspond to the radius (diameter) of the cone section by the plane running through the fix object. The fix object can be a point object or a planar object perpendicular to the cone axis.

Spherical Face

Spherical face diameter dimension is created in the plane running through the center of the sphere and the point where the face was selected (Fig.2.9.16). The initial plane position is chosen by the system. If necessary, you can change the dimensioning planes position by moving the handle **p1** with your mouse.

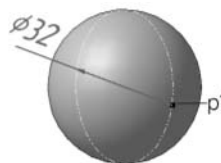


Fig. 2.9.16. Spherical face diameter dimensioning example

If the dimension position is fixed (see Section 2.9.2.7.1 on p. 644), following options of its position will be available:

- ▼ With reference to a point object— in the plane running through the sphere center and the selected point object; the dimensioning plane position will be determined by the system.
- ▼ With reference to a planar object— in the plane running through the sphere center parallel to the selected flat object.

Toroidal face

The toroidal face diameter dimension is created in the plane perpendicular to the face axis (torus circular axis) and running through the point where the face was selected (Fig.2.9.17).

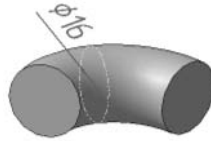
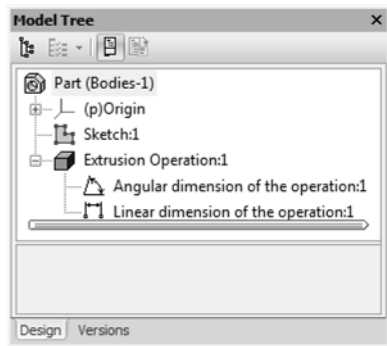


Fig. 2.9.17. Toroidal face diameter dimensioning example

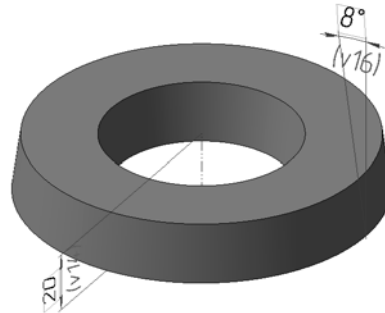
The fix object (see Section 2.9.2.7.1 on p. 644) can be any point object or a flat object perpendicular to the torus circular axis.

2.9.2.8. Derivative dimensions

Dimensions of sketches and dimensions of operations can be represented in the form of layout elements. As a result, **derivative dimensions** corresponding to dimensions of sketches and operations will appear in the Model Window. In the Design Tree, derivative dimensions are displayed as subordinate to sketches and operations (see Fig. 2.9.18).



a)



b)

Fig. 2.9.18. Derivative dimensions:
a) in the Design Tree, b) in the Model Window

In order to present dimensions of sketches and operations as layout elements, perform the following steps.



1. Call the **Operations — Layout Elements — Derivative Dimensions** command. The respective command calling button is located on the panel **Layout Elements**.
2. In the Design Tree, select objects the dimensions of which should be displayed as layout elements or specify any objects' primitive in the Model Window. a face, edge, or vertex.



Objects may be also selected before the command selection.

Specified objects are highlighted in a certain color and their names appear on the **List of Objects** bar. To cancel object selection, re-select it.

3. Enable the **Dimensions with zero value** option if you need to create derivative dimensions for dimensions of operations with zero value.



4. In order to change objects the dimensions of which should be displayed as layout elements, click the **Re-select** button on Special Control Panel and repeat object selection.
5. Having finished selection of objects, click the **Create Object** button on the Advanced Control panel.



Dimensions of the sketch with no created variables cannot be presented as layout elements.

Derivative dimensions of selected objects appear in the model window or in the Design Tree. The default color of dimensions displayed in the window is violet.



Derivative dimensions for dimensions with zero value are displayed only in the Design Tree. These derivative dimensions are not displayed in the model window.



Controlling dimensions of sketches and operations presented as layout elements retain their type, i.e. they remain the guiding ones. If the value of such a dimension is changed, the model will be rebuilt. The derivative dimension value is changed in the dimension value setting dialog (Figure 2.9.2.2.1 on p. 634).

The derivative dimensions can be transferred to a drawing when creating a model's associative views in it (see Section 3.6.1.2 on p. 1208). Dimensions transferred to the drawing are **projection designations**. Work with projection designations is described in Section 3.6.3.5 on p. 1226.

2.9.2.8.1. Specifics of Working with Derivative Dimensions

If a chamfer is built using a side and a 45° angle, a single linear derivative dimension will be created (in accordance with the standard). The dimension text of the dimension contains values of the chamfer's side and angle, for example, 5x45°. If necessary, you can create two derivative dimensions (linear and angular) for this chamfer. For this, do the following.



1. Select the derivative dimension of the chamfer in the Design Tree or in the model window and call the **Edit** command from the context menu.
2. Call the dialog for the dimension's text entry. For this purpose, click inside the **Text** field on the **Dimension** tab of the Property bar.
3. Clear the **Text after** field and close the dialog by clicking **OK**.
4. In order to finish editing dimensions, click the **Create Object** button on the Advanced Control panel. The dimension text of the edited dimension will contain only the chamfer side's value.
5. Call the **Derivative dimensions** command.
6. Select a chamfer in the Design Tree or in the Model Window:
7. Click the **Create Object** button. The angular derivative dimension will be added to the linear derivative dimension of the chamfer in the Design Tree or in the Model Window.

2.9.2.9. Editing derivative dimensions

You can edit the display of a derivative dimension: specify new drawing parameters and edit the dimension text.

Besides that, you can edit location of the specified derivative dimensions, i.e. specify the required plane (or a flat face) in which the dimensions will be positioned.

Editing of the derivative dimension display is described in Section 2.9.2.9.1 on p. 650; derivative dimensions positioning is described in Section 2.9.2.9.2 on p. 651.

2.9.2.9.1. Editing derivative dimension display

When editing the display of a derivative dimension, you can modify its dimension text and specify new drawing parameters.

In order to change displaying of the derivative dimension, select it in the Design Tree or in the Model Window and call the **Edit** command from the context menu.

Parameters, **Deleted**, **Properties** tabs will appear in the Property Bar.

Controls of the **Dimension** tab are described in Table 2.9.1, description of the **Parameters** tab can be found in Table 3.3.3 on p. 977.

Controls located in the **Properties** tab allow defining name of dimension in the Design Tree and color of drawing of dimensional lines.

Tab. 2.9.1. Control elements on the **Dimension** tab

Element	Description
Object	This field displays the name of object (sketch or operation), the dimensions of which are presented as layout elements (derivative dimensions).
Parameter	This field displays the name of the sketch's dimension or operation's parameter represented as a layout element (derivative dimension).
Length*	<p>The field allowing to define the length of derivative dimension leaders. By default, the field contains the value of the length of the leaders' dimension.</p> <p>To change default value, enter required length value from the keyboard or define it using the counter (the value may be both positive and negative).</p> <p>The leaders' length can also be changed using handles at the end points of the dimension line.</p> <p>When moving one of the handles using the mouse, the value of the length will be automatically defined and entered in this field.</p>
Text**	This is the field for calling the Enter Dimension Text dialog, in which you can specify the desired dimension value and set up its format (see Section 3.3.1.3 on p. 979).

Tab. 2.9.1. Control elements on the **Dimension** tab

Element	Description
Type***	The group of switches allowing to specify the type of the dimension line in a diameter dimension: <ul style="list-style-type: none"> ▼ full or with a break – for the diameter dimension; ▼ from center or not from center – for the radial dimension.

* It is available in the Property bar when editing a linear or angular dimension.

** It is available in the Property bar when editing a linear, diameter or radial dimension.

*** It is available in the Property bar when editing a diameter or radial dimension.



In order to finish editing the derivative dimension, click the **Create Object** button on the Advanced Control panel.



You can cancel dimension editing by pressing <Esc> or by clicking the **Cancel Command** button.

2.9.2.9.2. Locating the derivative dimensions

Immediately after their creation, the derivative dimensions are positioned as follows:

- ▼ the derivative dimension, that corresponds to the sketch's dimension, is positioned in the sketch's plane,
- ▼ the derivative dimension, that corresponds to the operation's dimension, is positioned in the plane, which is automatically defined by the system depending on the type of dimension and of the object created by the operation.

The plane in which a derivative dimension is located immediately after its creation is called the **initial plane**.

You can position one or several derivative dimension in a different plane by specifying the **base plane**. The following can be used as a base plane:

- ▼ Coordinate or auxiliary plane
- ▼ planar face.

The derivative dimension changes its position (it is positioned in the base plane), if the positioning conditions are met (see Section 2.9.2.9.3 on p. 651). If the conditions are not met, the derivative dimension will not change its position, and the respective message will appear on the screen.

If necessary, you can cancel changes after the dimension positioning in the base plane, and the dimension will be positioned in the initial plane.

Procedure for positioning of derivative dimensions is described in Section 2.9.2.9.4 on p. 652.

2.9.2.9.3. Prerequisites for positioning the derivative dimension in the base plane

- ▼ The derivative dimension is located in the base plane, if the plane is parallel to its initial plane.
- ▼ Besides that, the following positioning options are available for some types of dimensions:

- ▼ A linear dimension is positioned in the base plane, if the plane intersects its initial plane along the line which is parallel to the dimension line.
- ▼ The derivative dimension, corresponding to the *Wall Thickness* or *Slope Angle* extrusion operation's parameter, is located in the base plane, if this plane intersects the section and perpendicular to it in the intersection point.
- ▼ The derivative diametric dimension can be located in the base plane if the plane is perpendicular to the initial plane. In this layout, diameter dimension converts to linear.

2.9.2.9.4. Procedure for positioning the derivative dimensions

In order to change the plane, in which derivative dimensions are positioned, or cancel changes and position the dimensions in their initial planes, perform the following steps.



1. Call the **Editor — Position derivative dimensions** command. The respective command calling button is located on the **Layout Elements** bar.
2. Create a set of derivative dimensions, locations of which need to be changed. For this purpose, specify the required derivative dimensions in the Design Tree or in the Model Window:



Derivative dimensions can be selected before calling the command.

Specified dimensions are highlighted in a certain color and their names appear on the **List of Dimensions** bar. Reselecting a dimension cancels its selection.



- ▼ In order to select all derivative dimensions of the model at once, click the **Select all dimensions** on the **List of Dimensions** bar.



- ▼ To change dimensions, click the **Reselect** button on the Special Control Panel and re-select dimensions.



- ▼ In order to delete a dimension from the set, highlight its name in the list of dimensions on the **List of Dimensions** bar and click the **Delete** button.



3. Specify the plane for location of the set's dimensions using the **Base plane** group of switches.

- ▼ In order to locate the dimensions of a set in their initial planes, enable the **Autodefine** switch.



- ▼ In order to locate the dimensions of a set in a new plane, enable the **Specify manually** switch. Then, specify the required plane or a flat face in the Model Window. The specified object is highlighted by a certain color, and its name is displayed in the **Plane** field of the Property Bar.

In some cases, when positioning a derivative dimension there are a number of placement scenarios after a new plane is specified. An example is the placement of derivative dimension in the *Chamfer* operation when a new plane intersects the rib on which the chamfer is built and is perpendicular to it.



If there are several placement scenarios, the **Next object** and **Previous object** become available on the Advanced Control Panel. They are used to select the desired scenario.



4. Having set the parameters of locating dimensions, click the **Create Object** button on the Advanced Control panel.

Specified dimensions are located in accordance with the settings applied.

- ▼ If the plane for location of dimensions is selected automatically, each dimension of the set will be located in its initial plane, and the names of all dimensions of the set are deleted from the **List of Dimensions** bar.
- ▼ If the base plane is selected, dimensions of the set are located in the selected plane, lines of dimension projections and lines conditionally indicating objects with dimensions are displayed on the screen with connection lines. Names of dimensions are deleted from the **List of Dimensions** bar.

If, after clicking the **Create object** button, the names of dimensions remain on the **List of Dimensions** bar, it means that these dimensions cannot be located in the selected plane. The respective message will be displayed, and dimensions will remain in their current base planes (or initial planes). These dimensions will remain highlighted. You can assign another base plane to them.

By calling the **Locate derivative dimensions** command once, you can change location planes for any number of dimensions.



You can cancel locating dimensions by pressing <Esc> or by clicking the **Cancel Command** button.



Note that connection lines of dimensions are displayed only if the **Connection Lines** option is enabled in the dialog for dimension and designation display setup (see Section 9.1.11.12 on p. 1736).

2.9.3. Designations

The system allows to attach the following symbols:

- ▼ surface finish symbol,
- ▼ Base surface symbol
- ▼ Auxiliary extension line
- ▼ Stamping symbol leader
- ▼ Marking symbol leader
- ▼ Item symbol leader (only in assembly models),
- ▼ Form tolerance symbol.

The symbols in the model have the following features.

- ▼ The symbols get an associative relation to the objects selected during their creation. This relation causes each symbol to relate to a particular object in the model at all times.
- ▼ Every symbol exists in a plane that is called **symbol plane**. The symbols always remain within their planes, i.e. whenever the model is rotated, they will rotate correspondingly. If the model is rotated so that the symbol plane becomes perpendicular to the screen plane, then the symbol will not be displayed.
- ▼ If the symbol plane is not perpendicular to the screen, the symbol will always be displayed so that its text can be read from left to right.

To insert a symbol, use the appropriate symbol command, select the object(s) to which the symbol should be attached in the Model window, and adjust the symbol settings.

Common symbol creation methods are described in Section 2.9.3.1, while special techniques for creating particular symbols are described in Sections 2.9.3.2– 2.9.3.8.



For surface finish symbols and datum symbols, automatic creation is available. Automatic creation is on by default, which is indicated by a depressed **Object Autocreation** button on the Advanced Control Panel.



For other symbols, you will have to confirm their creation manually using **Create Object** button.

The automatic and manual object creation is detailed in Section 1.4.2.10 on p. 93.

If necessary, you can insert more than one symbol of the same type in one insert symbol command. To do this, without exiting the command, select the objects one by one which the symbols should be attached to.

The created designations will appear in the model window, while their icons will appear in the Design Tree.



To finish current command, click **Interrupt Command** button on the Advanced Control Panel.



To rotate the model so that the new symbol plane is parallel to the screen plane, without interrupting the insert symbol command. Use **Normal To...** command in **Orientation** button menu (see Section 2.1.3.2 on p. 121)



The 3D model operation mode doesn't support vector fonts including the KOMPAS-3D supplied fonts GOST type A (plotter), GOST type B (plotter), Symbol type A (plotter), Symbol type B (plotter) therefore you shouldn't use them for text included into designations.

Symbols can sometimes disturb from viewing the model. To hide a symbol, select it and use **Hide** context menu item.

Hidden symbols will not be transferred to the drawing when associative views of the model are created in it.

To make a hidden symbol visible, select it in the Design Tree and use **Show** context menu item.

You can hide all symbols at once using **View— Hide— Symbols** command. This will prevent both existing symbols and new symbols in the model from being displayed. To make all symbols visible, use the same command again. This will not display those symbols that were hidden individually.



If the element which the symbol was attached to is removed from the model in the process of editing (e.g. the edge to which the leader was attached is filleted), the symbol remains where it was. Therefore, to prevent such "loose" symbols from appearing in the model, it is recommended that the symbols are inserted only once the model shape is finished.

KOMPAS-3D uses the default symbol settings that are most commonly used in design documentation. If necessary, you can change these values. Parameters of designations setup is described in Section 9.2.7.3 on p. 1839.

2.9.3.1. Common Symbol Creation Methods

2.9.3.1.1. Specifying Symbol Plane Position

The symbol plane is parallel to the base plane. One of the coordinate planes is used as a base plane.

Initially, the base plane is selected automatically by the system depending on model position with respect to the screen plane. If necessary, you can select another base plane or move the symbol plane parallel to itself.

To select another base plane, choose the desired plane in the **Base Plane** drop-down list on the **Symbol** tab of the Property bar (Fig. 2.9.19). You can also use context menu item **Set Plane XY/ZX/ZY to the base plane**.

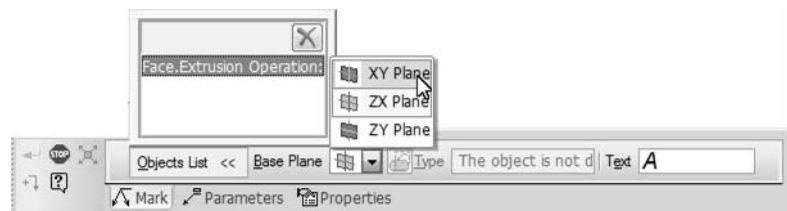


Fig. 2.9.19. Property bar during leader construction

By default, the symbol plane will run through the point where the object which the symbol is attached to was selected (or through the object itself if it is a point object). If necessary, you can move the symbol plane from its default position.



When creating a surface finish symbol or a datum symbol, this opportunity will only be available if automatic object creation is disabled and only once the symbol is generated (symbol insertion point is specified for a surface finish symbol or the point determining the position of the text box and the starting point of one branching is specified for a datum symbol).



To move the symbol plane, click **Position** button on the **Symbol** tab of the Property bar. The ghost of the designation symbol's plane is displayed.

Symbol plane offset can be set freely or fixed to match an object existing in the model.

- ▼ For free positioning of the symbol plane, move its ghost's handle with your mouse. The ghost will move so that the symbol plane remains parallel to the base plane and all branching starting points remain on the object(s) which the symbol is attached to.



If the symbol plane cannot be moved (e.g., because the object which the symbol is attached to is either a point or an edge parallel to the base plane), the ghost will remain in its place.

- ▼ To fix the symbol plane position, select the object to fix to. The symbol plane will run through that object. The fix object can be any point or straight or planar object in the model that is in such position that makes it possible to draw a plane through it parallel to the base plane and

crossing the object which the symbol is attached to. Selecting the fix object again will cancel symbol plane position fixing.

An associative relation will be established between the symbol plane and the fix object. This relation causes the symbol plane to follow the fix object wherever it is moved.



If the symbol plane is fixed, then it will be impossible to select another base plane (base plane list will be unavailable). You will need to unfix the symbol plane to be able to select another base plane.

2.9.3.1.2. Adding and Removing Branchings

Leaders, marking and stamping symbols, ref.no. leader and geometric tolerances can have more than one branching.

All branchings of the same symbol shall belong to that symbol's plane, i.e. points belonging to the symbol plane shall be specified as the second and subsequent starting points of the branching.

When specifying start points of additional branches, when the cursor passes above a body (or a surface), intersecting with the symbol plane, the line of intersection of the body (plane) with the symbol plane is displayed as a dashed line. Faces of solids and surfaces can only be selected at points belonging to that line. Whenever the mouse cursor is hovered over this line, the *Point on Line* snap name and a face icon will be displayed.

To add the branching to the symbol, move the cursor to the required point of this branching start (on the line of intersection of the face and symbol plane) and click the mouse, when the *Point on Curve* snap name appears beside it.



The starting point of a branching can be a 3D point in the symbol plane, as well as a point belonging not to a face, but, for example, to a 3D line, auxiliary axis, etc. An indicator of whether the point can be the starting point of a branching is the *Nearest Point* snap name and the icon of the object which this point belongs to appearing next to the mouse cursor when it hovers over this point.

The symbol is considered related to an object if the symbol's branching starting points belong to that object. List of objects which the symbol relates to can be found in the **Objects List** pane on the **Symbol** tab of the Property bar (see Fig. 2.9.19 on p. 655).

When an object is selected in the list, its corresponding branching is highlighted in the Model window.



To delete a branching, select it in the list and press *Delete* or click **Delete** button on the **Objects List** toolbar.

2.9.3.2. Surface Finish

The surface finish symbol can be attached to the following objects:

- ▼ solid or surface face,
- ▼ solid or surface edge,

- ▼ 3D polyline segment,
- ▼ dimension,
- ▼ marking.



To create the symbol use the **Face Finish** command.

Select the object which the surface finish symbol will be attached to.



To select another object, click **Select Other** on the Special Control Panel and select the desired object.

After calling the command, a designation ghost will appear. The ghost will appear in the symbol plane. If necessary, you can move the symbol plane (see Section 2.9.3.1.1).



If a dimension or a symbol is selected as the object which the surface finish symbol should be attached to, then the surface finish symbol will be created in that object's plane. You will not be able to change the plane's position.



By default a surface finish symbol is created, whose machining type is not specified. Also, **Without Machining Type** option will be enabled in the **Type** group. To create a finish symbol for a surface generated with or without material removal, set the appropriate option in this group.



Enter the text and configure symbol drawing (see Section 3.3.7.3.1 on p. 1022 and Section 3.3.7.3.2 on p. 1023).

By default, surface finish symbol will be created on leader shelf.

Select the start point of Leader Bend.

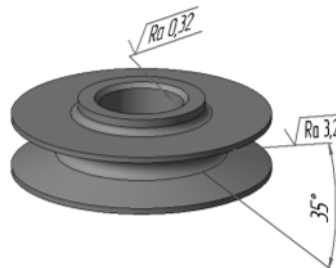


Fig. 2.9.20. An example of applying a finish symbol

The structure of the surface finish symbol is defined by GOST2.309-73 (see Fig.3.3.48 on p. 1021).

You can select the structure used in the current model during surface finish symbol setup in the current model (see Section 9.2.7.3.12 on p. 1842). By default, in new models designations of finish are created according to the change in GOST2.309–73.



Without leaving the command, you can create several surface finish symbols. Selected symbol type and the generated text will be saved.

2.9.3.3. Datum

Datum symbols can be attached to following object types:

- ▼ faces of surfaces and bodies,
- ▼ Edges of surfaces and solids
- ▼ dimensions,
- ▼ Auxiliary axes and planes,
- ▼ 3D Curves.



You can create datum surface symbols using **Datum** command.

Select a surface target for a datum symbol.



To select another object, click **Select Other** on the Special Control Panel and select the desired object.

Once the object is selected, a symbol ghost will appear. The middle point of the base of the triangle designating the datum will match with the point at which the object was selected. The ghost will appear in the symbol plane. If necessary, you can move the symbol plane (see Section 2.9.3.1.1).



If a dimension or a symbol is selected as the object which the datum symbol should be attached to, then the datum symbol will be created in that object's plane. You will not be able to change the plane's position.



By default, a datum symbol, perpendicular to the selected reference object, is created. Also, **Perpendicular to Base Object** option will be enabled in **Type** group on the Property bar. If a pitched symbol should be created, set the **Arbitrary Orientation** option.



In the **Text** field on the Property Bar, the letter is displayed that is suggested by the system as the Enlarged Detail.

By default, **Auto-sorting** option is on: in this way all datum symbols are assigned alphabetical characters automatically as they are created, thereby preventing duplicate designations. If **Auto-sorting** is on, **Text** field will be unavailable. If you need to enter the datum character manually, disable the **Auto-sorting**. **Text** field will be enabled; however, duplicate character control will be deactivated.



If **Auto-sorting** is then re-enabled, the manually entered characters will be replaced with automatic characters.

Select the point defining the text frame position.

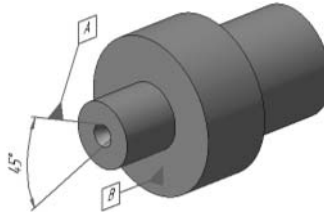


Fig. 2.9.21. An example of applying a datum symbol

2.9.3.4. Leader

Leaders can be attached to following object types:

- ▼ solid or surface face,
- ▼ coordinate plane,
- ▼ auxiliary (construction) plane,
- ▼ sketch point,
- ▼ 3D point,
- ▼ 3D line vertex,
- ▼ a sketch line,
- ▼ 3D polyline segment,
- ▼ a 3D curve,
- ▼ solid or surface edge,
- ▼ coordinate axis,
- ▼ auxiliary axis,
- ▼ dimension,
- ▼ marking.



You can create a custom leader using **Leader** command.

Select the object which the first branching of the leader will point at. The first branching will start from the point at which the object was selected.

Select the start point of a leader shelf.

The symbol ghost will appear. The ghost will appear in the symbol plane. If necessary, you can move the symbol plane (see Section 2.9.3.1.1 on p. 655).



If a dimension or a symbol is selected as the object which the leader should be attached to, then the leader will be created in that object's plane. You will not be able to change the symbol plane's position.

To specify starting points for additional branchings, select the points belonging to the symbol plane. For details on how to add and remove branchings, see Section 2.9.3.1.2 on p. 656.

Enter the text and configure leader drawing (see Section 3.3.7.4.1 on p. 1024 and Section 3.3.7.4.2 on p. 1025).



Edit Points button on the Advanced Control Panel allows changing the leader configuration (see Section 3.3.7.4.3 on p. 1027).

To fix the symbol, click **Create Object** button.

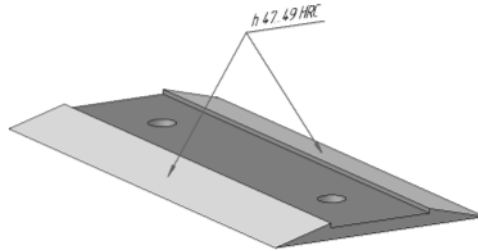


Fig. 2.9.22. Example of creating a leader

2.9.3.5. Stamping Symbol



To attach stamping symbol you can use faces of bodies and surfaces.

You create a leader for a stamping symbol using **Stamping Symbol** command.

Select the object which the first branching of the leader will point at. The first branching will start from the point at which the object was selected.

Specify the point determining the stamping symbol position.

The symbol ghost will appear. The ghost will appear in the symbol plane. If necessary, you can move the symbol plane (see Section 2.9.3.1.1).

To specify starting points for additional branchings, select the points belonging to the symbol plane. For details on how to add and remove branchings, see Section 2.9.3.1.2 on p. 656.

In the **Text** field on the Property Bar the text is displayed that is suggested by the system as the Stamping Symbol. If needed, you can change the contents and the display style of the text (see Section 3.3.7.5.1 on p. 1030).

Configure stamping symbol drawing (see Section 3.3.7.5.2 on p. 1031).



Edit Points button on the Advanced Control Panel allows changing the leader configuration (see Section 3.3.7.4.3 on p. 1027).



To fix the symbol, click **Create Object** button.

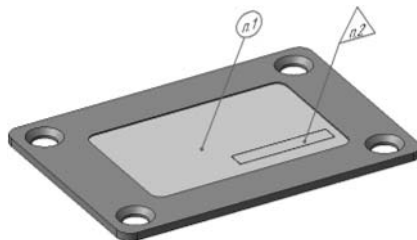


Fig. 2.9.23. An example of applying a stamping mark

2.9.3.6. Marking Symbol



Faces of bodies and surfaces may be used for marking symbols.

You can create a leader for a marking symbol using **Marking Symbol** command.

Select the object which the first branching of the leader will point at. The first branching will start from the point at which the object was selected.

Specify the point determining the marking symbol position.

The symbol ghost will appear. The ghost will appear in the symbol plane. If necessary, you can move the symbol plane (see Section 2.9.3.1.1).

To specify starting points for additional branchings, select the points belonging to the symbol plane. For details on how to add and remove branchings, see Section 2.9.3.1.2 on p. 656.

In the **Text** field on the Property Bar the text is displayed that is suggested by the system as the Marking Symbol. If needed, you can change the contents and the display style of the text. This is done in the same way as when creating a stamping symbol in a graphic document (see Section 3.3.7.5.1 on p. 1030).

Using the **Arrow** list on the **Options** tab of the Property bar you can select the type of the arrow of the leader line. If this arrow type must be used in the Marking Symbols up to the end of the current working session, enable the option **By default**.



Edit Points button on the Advanced Control Panel allows changing the leader configuration (see Section 3.3.7.4.3 on p. 1027).



To fix the symbol, click **Create Object** button.

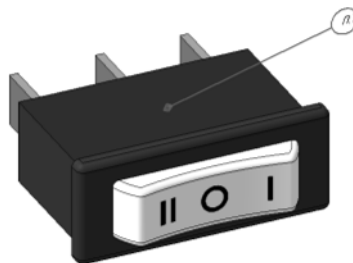


Fig. 2.9.24. An example of applying a marking symbol

2.9.3.7. Ref.No. Leader

The following objects can be used to designate a position:

- ▼ solid or surface face,
- ▼ solid or surface edge.



To create a leader line to designate Ref.No. call the **Ref. No.Designation** command.

Select the object which the first branching of the leader will point at. The first branching will start from the point at which the object was selected.

Select the start point of a leader shelf.

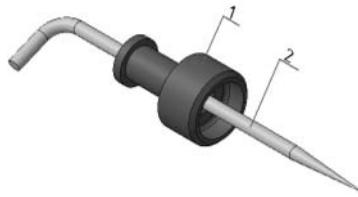
The symbol ghost will appear. The ghost will appear in the symbol plane. If necessary, you can move the symbol plane (see Section 2.9.3.1.1).

In the **Text** field on the **Symbol** tab of the Properties bar, the system-generated text is suggested — the reference number. If needed, you can change the number and its display style, as well as create additional shelves with reference numbers. This is done in the same way as when creating a position symbol in a graphic document (see Section 3.3.7.7.1 on p. 1033).

To specify starting points for additional branchings, select the points belonging to the symbol plane. For details on how to add and remove branchings, see Section 2.9.3.1.2 on p. 656.

Edit Points button on the Advanced Control Panel allows changing the leader configuration (see Section 3.3.7.4.3 on p. 1027).

To fix the symbol, click **Create Object** button.



An example of applying a position reference number.

2.9.3.8. Geometric Tolerance

Geometric tolerance symbol can be attached to the following objects:

- ▼ solid or surface face,
- ▼ coordinate plane,
- ▼ auxiliary (construction) plane,
- ▼ solid or surface edge,
- ▼ 3D polyline segment,
- ▼ auxiliary axis,
- ▼ dimension,
- ▼ marking.



You can create a geometric tolerance symbol using **Geometric Tolerance** command.

Select the object which the symbol will be attached to.

Select the point that will determine the marquee box position. By default, the bottom left corner of the border is placed in the selected point. Also, in the **Anchor Point** list, the **Bottom Left** option is selected. To change the position of the frame relative to the insertion point, expand the specified list and select the desired line.

To make the box appear vertical, turn on **Vertical** option on the Property bar.

The marquee box ghost will appear. The ghost will appear in the symbol plane. If necessary, you can move the symbol plane (see Section 2.9.3.1.1).



If a dimension or a symbol is selected as the object which the symbol should be attached to, then the geometric tolerance symbol will be created in that object's plane. You will not be able to change the symbol plane's position.

Generate the tolerance table (see Section 3.3.7.13.1 on p. 1046).



To create a branching with an arrow or a triangle, click the appropriate button on the Special Control Panel and select the branching starting point in the symbol plane. End point of all branches is the basic point (point of the frame in **Basic Point**) list.



To finalize a branching creation, click the respective button on the Special Control Panel.

For details on how to add and remove branchings, see Section 2.9.3.1.2 on p. 656.



To create the next branching click the appropriate button again.

Edit Points on the Special Control Panel makes it possible to adjust the configuration of branchings. Once it is clicked, the points will be displayed as black squares and circles. Move the cursor to any handle. Shape of the cursor will change: it will turn into a four-sided arrow. You can move the handles by dragging them with your mouse.



To exit the mode of editing the handle positions, release the **Edit Points** button.

To fix the designation, click **Create Object** button.

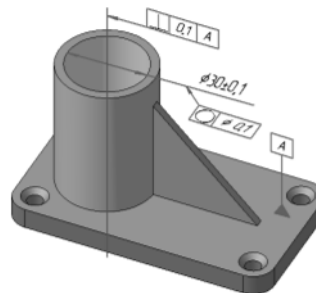


Fig. 2.9.25. Example of entering of geometric tolerance and location of surface

2.9.4. Cosmetic Thread

In KOMPAS-3D models, threads are created not as helical surfaces, but as symbols: wireframe cylinders or cones. To build a symbol of thread in a model, perform the following actions.



1. Call the command **Cosmetic Thread**.

On the Properties Bar controls will appear for setting thread parameters.

2. Specify the base item of the thread: a cylindrical or conical face.

A ghost of a symbolic thread depiction will appear in the model window.

The starting border of the thread is determined automatically as you specify the base object. If you need to change starting border, activate the **Initial Border** switch, then specify on the Design Tree the surface, face, or plane for which you want to plot the thread.

3. Set thread standard, diameter and pitch. This is done the same way, as when building a threaded hole (see p. 3 and 4 Section 2.3.9.1.4 on p. 212).

For non-standard threads, the **Auto Define Thread Diameter** option is available. Enable the automatic diameter determination option using the dimensions of the object to be threaded. The automatically calculated diameter value is displayed in the field **Thread Diameter**. Set thread pitch.

4. Set thread length. To do this, use the **Thread length** group of switches. The following methods are available:



- ▼ **To specified length:** thread length is defined by the specified distance. Enter thread length value in the **Thread Length** field.



- ▼ **To finish border:** thread length is determined automatically using the end limit of the thread. If you need to automatically change a certain thread end limit, activate the **Final Border** and specify the desired surface or rim.



As a thread starting/end limit, a rim of the thread reference object face or a rim of a co-axial conical (countersink surface) or toroidal (rounding surface) face must be specified. The edge that can be used is highlighted when the mouse pointer hovers over it.

Turn on the **Fitting** option if you need to change the geometry of the base object of the thread (see Section 2.9.4.1 on p. 665).

5. If necessary, use the following additional functionalities.



- ▼ Copying existing thread parameters. For that, call the command **Thread by sample** by clicking the corresponding button on the Advanced Control Panel or from the context menu. Select the thread to copy parameters from in the model window or on the Design Tree. The thread being created will have the same parameters as the template thread. Edit the parameters if necessary.



- ▼ Cancel selection of all objects. To do this, click the **Select other** button on the Advanced Control Panel.



6. Having finished adjustment, click **Create Object** button to fix the cosmetic thread in the model.



The Model window will show a wireframe cylinder or cone (Fig. 2.9.26) representing the tread, and the Model Tree will show a thread icon. Next to the cosmetic thread on the Tree, thread designation and a *Symbolic depiction message* (for non-standard thread a *Symbolic thread depiction message* is used).

If 'fitting' was used when creating symbolic thread depiction, its icon appears on the Model Tree as a subordinate object for the thread depiction. You can turn on/off 'fitting' without entering thread editing mode using the command **Fitting** on the context menu of the thread on the Model Tree.



Building symbolic depiction of standard conical thread has the following special features:

- ▼ Creating an object is possible only by using fitting.
- ▼ Thread can be plotted only for a given depth. Maximum depth depends on the hole dimensions and is set by the standard.

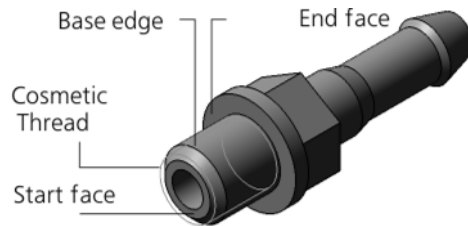


Fig. 2.9.26. Cosmetic Thread on Nipple part



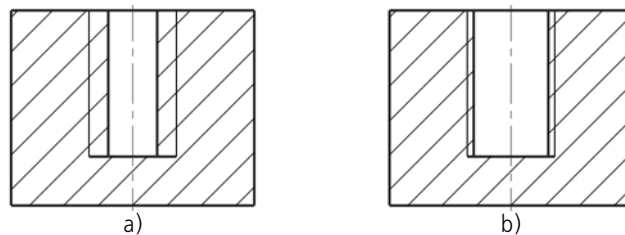
Note that a cosmetic thread is always shown in a model in its entirety. For example, suppose that the **Trim by Surface** command cuts off or slashes through a feature which bears a thread. In this case, the cosmetic thread will be displayed in the model in its entirety, even though the surface (or a portion thereof) that bears the thread is not shown.

When creating associative model views in drawings, which contain cosmetic threads, the latter can also be drawn on the views (see Section 3.6.1.3.3 on p. 1213).

2.9.4.1. Using fitting

Fitting is used if you need to change the geometry of the base object of the thread to match the selected nominal diameter. In the process, for a cylindrical surface – the diameter is 'adjusted' to match thread parameters (Fig. 2.9.27), for a conical surface – diameter in the main plane, and the cone angle.

Turning fitting on and off is done using the **Fitting** option on Properties Bar, when creating or editing cosmetic thread.

Fig. 2.9.27. Example of a thread in a simple hole:
a) without fitting, b) with fitting

When fitting a hole with a countersink (chamfer or rounding) the depth of countersink is changed. To 'fit' the depth of countersink, enable additional fitting switch **Offset 1**. This allows offsetting the starting limit of the threaded section to the initial position and keep the initial depth of the countersink (Fig. 2.9.28).

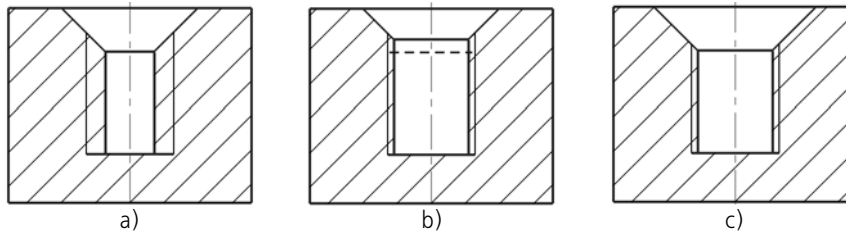


Fig. 2.9.28. Example of a thread in a countersinked hole:
 a) without fitting, b) with fitting, **Offset 1** disabled (the dashed line designates the depth of countersink prior to fitting), c) with fitting, **Offset 1** enabled



When fitting a hole with the non-flat end face, the length of threaded section changes. To 'adjust' this length, enable the additional fitting switch **Offset 2**. This allows offsetting the end limit of the thread reference object leaving unchanged the length of the section of the surface, on which the thread is created, see Fig2.9.29.

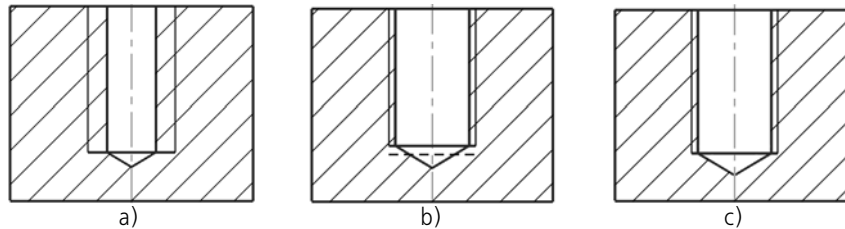


Fig. 2.9.29. Example of creating a thread in a hole with non-flat end face:
 a) without fitting, b) with fitting, **Offset 2** disabled (the dashed line designates the depth of countersink prior to fitting), c) with fitting, **Offset 2** enabled



For a rod with external thread, with a chamfer or rounding at the starting/end limit of the thread, additional fitting is done similar to a hole.

2.10. Tolerances. Model recalculation based on tolerances

2.10.1. General Information

All objects in the model are created by performing operations. If numeric parameters of an operation are expressed in linear or angular values, tolerances may be assigned for the parameter values. In addition, you may assign tolerances for sketch dimensions. For more information about dimensions of sketches and operations, see Section 2.9.2.2 on p. 632; for tolerances and their purpose, see Section 2.10.2 on p. 667.

When working with a model, you can use **model dimensions recalculation mode**. This mode is used to recalculate model dimensions taking into account the assigned tolerances. Model dimensions calculation is done in accordance with the current recalculation.

KOMPAS-3D supports three system recalculations of dimensions: **to middle part of the tolerance field, by upper limit and by lower limit**. It is also possible to create **user** recalculations.

For more information on model dimensions recalculation mode and working in this mode see Section 2.10.3 on p. 671.

2.10.2. Tolerances

KOMPAS-3D supports assigning tolerances for sketch dimensions and values of numeric parameters of an operation for which the following conditions are met:

- ▼ the parameter is expressed in linear or angular values;
- ▼ the parameter variable is not an information variable (for details on informational variables see Section 7.1.1.2.1 on p. 1570).

It is possible to assign **general** and **individual** tolerances.

A tolerance is assigned automatically based on settings (see Section 9.2.7.2.10 on p. 1838). These settings determine:

- ▼ objects, dimensions, and parameters that should have the tolerance;
- ▼ the value of the individual tolerance for dimensions of different types: linear, angular, diameter, radial; dimensions that no value of an individual tolerance has been assigned to have a general tolerance.

Conditions for using a general tolerance in a model and its setup are described in Section 2.10.2.1 on p. 668.

If necessary, when creating or editing an object the user can manually enable/disable a tolerance for a dimension (parameter), modify the value of the individual tolerance, change a general tolerance to an individual one and back (see Section 2.10.2.2 on p. 669).

The individual tolerance is displayed both in the Variables window and in the dimensional text of the sketch dimensions and the dimensions corresponding to the operation parameter (for information on the dimensions of sketches and operations, see Section 2.9.2.2 on p. 632; on

tolerances in the Variables window, see Section 7.1.4.2 on p. 1587). General tolerances are shown in the Variables window.



If the model requires a rebuild, the Variables window may display incorrect maximum values of parameters or dimensions calculated using the common tolerances. To view the correct values, rebuild the model.

Figure 2.10.1 shows an example display of individual tolerances in sketch dimensions and operation dimensions during extrusion creation.

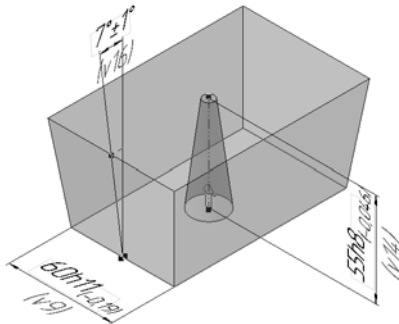


Fig. 2.10.1. Displaying individual tolerances during extrusion creation



Tolerances can only be assigned to parameters of those operations that have been created directly in the current model (for example, for parameter values of a cut extrusion or mating element, etc). Tolerances for parameter values of operations performed in a component must be assigned when working with this component.

Common and individual tolerances used in the model **dimensions recalculation mode**. More on the mode, see Section 2.10.3 on p. 671.



Individual tolerance can be assigned for control and information sketch dimensions (with or without a variable). However, the model dimensions recalculation mode uses only the controlling sketch dimensions that have a variable.

2.10.2.1. General tolerances

General tolerances in KOMPAS-3D are set for four precision classes based on GOST 30893.1-2002 (ISO 2768-1-89).

Enabling the use of common tolerances and selection of the desired accuracy class is done in the general tolerances configuration dialog (see Section 9.2.7.4.9 on p. 1854). By default, the use of common tolerances is On.

A general tolerance is assigned automatically for all dimensions and parameters that can have a tolerance, except for:

- ▼ a dimension or parameter with an individual tolerance or the sign *Nominal (framed)* (see Section 2.10.2.2 on p. 669 for more information),

- ▼ an angular parameter of a 3D curve, point or construction plane.
- ▼ a linear dimension or a linear parameter whose absolute value lies in the range of [0...0.5),
- ▼ angular parameters with the value of 0,
- ▼ angular parameters of a rotation operation with the value of 360.

Setting up the maximum deviations for parameter values and dimensions on common tolerances is performed using data contained in the **deviations file** (see Section 2.10.2.1.1 on p. 669).



According to the standard, maximum tolerances for blunt edge parameters differ from other linear parameter deviations for the same value range and accuracy class. In KOMPAS-3D, common tolerances for all linear parameters are assigned using the same table of standards. You can assign the desired individual tolerances for blunt edge parameters.

2.10.2.1.1. Deviations file

Maximum deviations of dimensions and parameter values for general tolerances are extracted from the **deviations file**. The file name and location is determined by the GENERALTOLERANCE variable of the environment KOMPAS-3D (see Section 9.5.4 on p. 1886). By default, the deviations file is named as *Gen_tol.ttl* and located in the \Sys subfolder of the KOMPAS-3D main folder. Use *Kompas.ini* (see Section 9.5.3 on p. 1883) if the name and/or position of the deviations file should be different from the default values.



If the deviations file is absent in the path indicated by the GENERALTOLERANCE variable or *Kompas.ini* general tolerances are not assigned to dimensions of sketches and parameter values of operations.

2.10.2.2. Tolerance assignment

To assign or modify tolerance for a sketch dimension or an operation parameter value, the **Assign tolerance** dialog is used (Fig. 2.10.2).

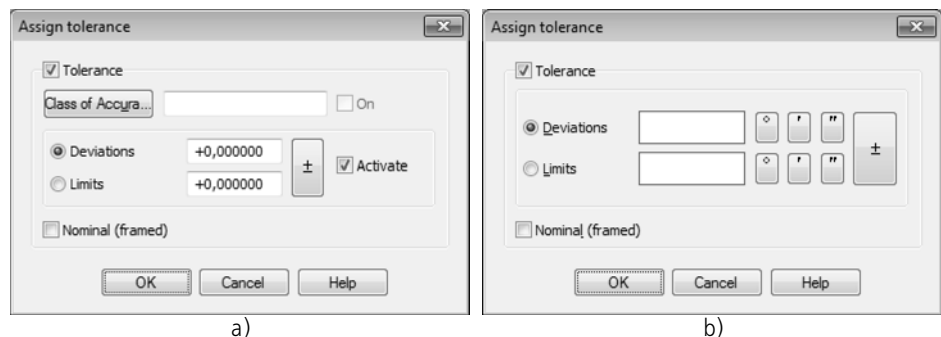


Fig. 2.10.2. Assign tolerance dialog:
a) for a linear value, b) for an angular value

The controls of this dialog are described in Table 3.3.5 on p. 982.

For new dimensions and operation parameters:

- ▼ the state of the **Tolerance** option (enabled/disabled) and the default tolerance value are determined by setting up default tolerances (see Section 9.2.7.2.10 on p. 1838),
- ▼ the state of the following controls: **Deviations, Limits, Enable** is determined by setting up tolerance parameters and tolerance ranges (see Section 9.2.6.10.10 on p. 1779).

The dialog can be accessed both in the course of the operation and out of the process. For more details on calling the dialog, see Section 2.10.2.2.1 on p. 670.

The main actions performed in the dialog are:

- ▼ **Individual tolerance assignment**
Fill in the required dialog fields and click **OK**.
- ▼ **Common tolerance assignment**
Clear all the fields of the dialog and click **OK**.
Common tolerance is assigned for a sketch dimension or an operation parameter value if the **Use Common tolerances according to GOST 30893.1 (ISO2768-1)** option is on in the common tolerances configuration dialog (for information on configuring common tolerances see Section 9.2.7.4.9 on p. 1854). If the option is disabled, no general tolerance will be assigned; the dimension or the parameter value will have an individual tolerance or no tolerance.
- ▼ **Disabling the ability to assign tolerance** (common or individual)
Turn off the **Tolerance** option and click **OK**.



If deviation limiting variables have been created for a sketch dimension or operation parameter (see Section 7.1.5.2 on p. 1592), and the **Expression** field of the Variables window is filled for at least one of them, the following controls will not be available in the dialog: the **Class of Accuracy** button, the field for class of accuracy setting, the option controlling displaying of the class of accuracy, as well as the fields of limit deviations/values of the dimension.



The **Assign tolerance** dialog can be used to modify **the display** of a dimension text for the operation's dimension, for example, to enable showing maximum dimension values instead of deviations. Changes so made are saved before the operation is completed. When the operation is edited again, its dimensions appear based on the setup of tolerance and tolerance range parameters (see Section 9.2.6.10.10 on p. 1779).

2.10.2.2.1. Displaying the Assign tolerance dialog

The **Assign tolerance** dialog (see Fig. 2.10.2 on p. 669) dialog may be accessed both in the course of an operation and out of the process.



In the course of an operation, the dialog appears on the screen after clicking the **Assign tolerance** button. The button is located on the Property bar next to the operation's numeric parameter field.



The **Assign Tolerance** button is available in the Property bar if the **Tolerance Control Buttons** option is enabled in the Property bar display parameters setup dialog (see Section 9.1.3.6 on p. 1683).

When out of the operation process, the **Assign tolerance** dialog may be accessed from Variables window (see Section 7.1.4.2 on p. 1587).

The Assign tolerance dialog may be also called when editing a sketch or operation dimension by clicking the **Assign Tolerance** button in the dimension value setting dialog. For more information about dimension value setting, see Section 2.9.2.2.1 on p. 634.

2.10.3. Model dimensions recalculation mode

Model dimensions recalculation mode is a special KOMPAS-3D mode for working with three-dimensional models. This mode is used to recalculate model object and component dimensions taking into account the assigned tolerances (both common and individual, see Section 2.10.2 on p. 667). Model dimensions calculation is done in accordance with the current recalculation.

KOMPAS-3D supports three system recalculations of dimensions: **to middle part of the tolerance field, by upper limit** and **by lower limit**. It is also possible to create **user** dimension recalculations. For more information on recalculations of dimensions, see Section 2.10.3.1 on p. 672.

When you activate the recalculation mode, the following happens:

- ▼ the values of variables corresponding to the model dimensions to which tolerances are assigned are changed in accordance with the parameters of the selected recalculation;
- ▼ the recalculated values of variables are displayed in the **Recalculated Value** column of the Variables window;
- ▼ the following text is added to the header of the document window: “(Recalculation — <the name of the selected recalculation>)”;
- ▼ the model is rebuilt based on new variable values.

For example, system recalculation of dimensions is used for the model transformation by upper limit. When you enable the mode, all the dimensions involved in the recalculation take the highest limit values. The model is rebuilt using the new dimensions.



As model recalculation using tolerances actually means rebuilding the model with the new variable values, you should remember that:

- ▼ dimensions that do not affect the model (information dimensions or the dimensions that do not have a variable) do not change even if they have a tolerance;
 - ▼ to be rebuilt correctly, the model must be correctly parameterized (e.g., a sketch must be completely defined, etc.).
-



In the recalculation mode, the common tolerance settings configured and saved in component files are applied to model components. If the use of common tolerance is disabled for a component, the common tolerances are not applied to the dimensions of such a component when the recalculation mode is active.

Recalculation control is performed using the **Document manager** (see Section 2.10.3.2 on p. 673).



To enable the model dimensions recalculation mode, click the **Recalculate dimensions based on tolerances** on the **Modes** bar or call the command of the same name from the **Service – Recalculate dimensions** menu.



The **Recalculate dimensions based on tolerances** button is not available when editing a component in-place.

The **Recalculate dimensions based on tolerances** button menu contains: the list of all recalculations of the model dimensions and the **Recalculation Control...** and **Create Copy** commands. The commands for creation of new objects and for editing existing objects are unavailable if the recalculation mode is enabled. For more information on working in the mode see Section 2.10.3.3 on p. 679.

Using the **Save As...** command, the recalculated model can be saved in a different format.



If the model which is in mode recalculation was closed with saving, then upon opening the model its status will be the same as at the closing: recalculation enabled, and the dimensions of the model correspond to the current recalculation.

After disabling the mode, the model will return to the state which it had at the moment of entering the mode.

2.10.3.1. System and user dimension recalculations

Dimension recalculation is a collection of the following parameters:

- ▼ a set of variables corresponding to dimensions of graphic documents or operations (for more information on dimensions, see Section 2.9.2.2 on p. 632);
- ▼ a method to recalculate the value of each variable in the set (see Table 2.10.1).

System recalculations of model dimensions:

- ▼ to middle part of the tolerance field (the default option),
- ▼ by upper limit,
- ▼ by lower limit.

The set of variables for system recalculation is generated automatically. It includes all the variables of the model (including component variables) that are related to dimensions of sketches and operations. The values of the variables from this set are recalculated using one method.

For example, recalculation to the middle of the tolerance field is used to rebuild the model. When you enable the recalculation mode, the variables affected by the recalculation take the values corresponding to the middle of the tolerance field. The model is rebuilt using the new variable values.

Users can create their own dimension recalculations, called **user recalculations**. The set of variables for user recalculation is created manually. For each variable from the set, a method to recalculate its values is assigned:

- ▼ to the middle of the tolerance field

- ▼ by upper limit,
- ▼ by lower limit,
- ▼ using a factor.

For information on creating, editing, or deleting a user recalculation, see Section 2.10.3.2.1 on p. 676.

Methods for model dimensions recalculation and the corresponding recalculation formulas are given in Table 2.10.1. The letters in formulas have the following meanings:

A – the recalculated value of a dimension (of a sketch or operation),

N – a nominal value of a dimension,

ES – top deviation,

EI – bottom deviation.

Tab. 2.10.1. Model dimension recalculation methods

Dimension recalculation method	The formula for recalculation of dimension corresponding to the recalculation method
to the middle of the tolerance field*	$A=N+(ES+EI) \cdot 0.5$
by upper limit	$A=N+ES$
by lower limit	$A=N+EI$
using a factor**	$A=N+EI+(ES-EI) \cdot K$, where $0 < K < 1$

* It is used by default in the user recalculation.

** It is only used for variables on custom recalculations. It is unavailable for components.



If the sketch or operation dimension is an informational one, the dimension variable value is not recalculated (and remains unchanged).

2.10.3.2. Recalculation a control

Model dimensions recalculation is controlled in the **Document manager**. You can use it to perform the following actions:

- ▼ creation, editing, or deletion of user recalculations;
- ▼ selection of the recalculation to be used for calculation of model dimensions (i.e. making a certain recalculation as current).

Creation, editing, and deletion of custom recalculations are described in Section 2.10.3.2.1 on p. 676, the methods for working with recalculations — in Section 2.10.3.2.2 on p. 677.



Go to **Recalculation control...** to start dimension recalculation control. The command is in the menu of the **Recalculate dimensions based on tolerances** button in the **Modes bar** (see Fig. 2.10.5 on p. 680) and in the **Service — Recalculate dimensions** menu. The command is available outside the recalculation mode.

Upon calling the command, the **Document manager** dialog will appear (Fig. 2.10.3). The **Recalculate dimensions** element is selected and the **list of recalculations** is expanded in the left part of the dialog, in the Document Tree. The list of recalculations contains the names of all the model dimension recalculations, both system and user ones (if any). Letter 's' in parenthesis precedes each system recalculation. The current recalculation will be marked with a red check mark.

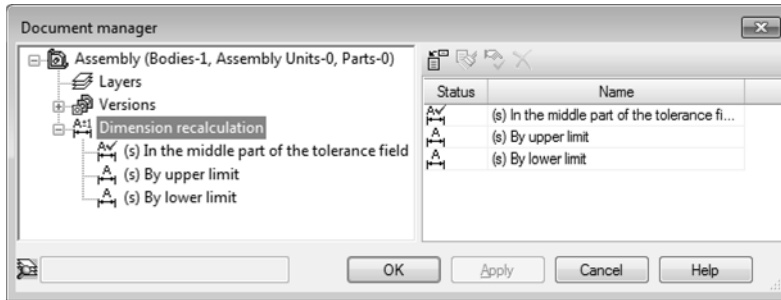


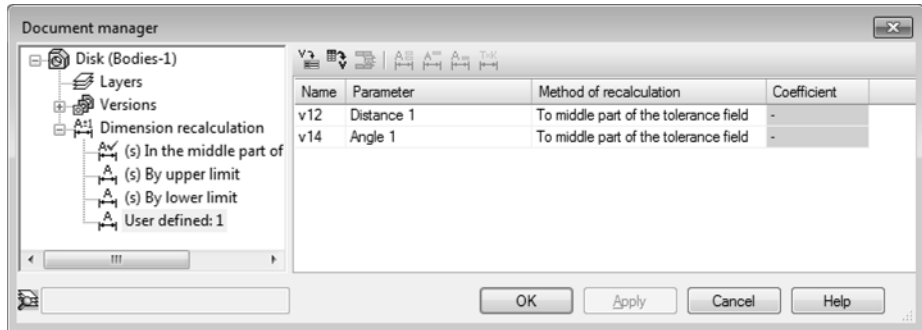
Fig. 2.10.3. Managing the recalculation mode in the Document manager

The right part of the dialog is intended for work with recalculations. Its contents depend on the action currently performed — recalculation control or custom recalculation setup.

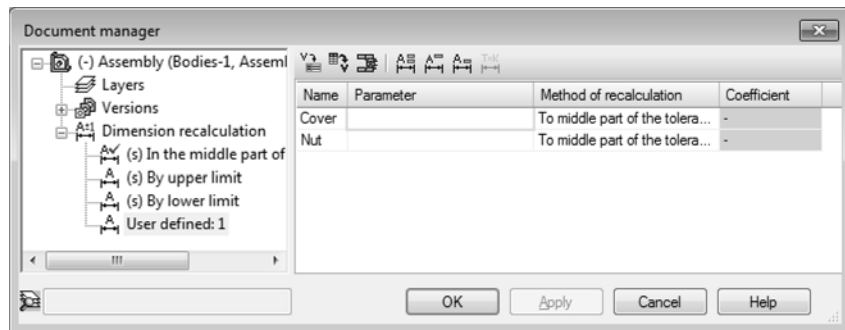
- ▼ Select the **Recalculate dimensions** in the Document Tree for recalculation control. The following appears in the right part of the dialog (Fig. 2.10.3):
 - ▼ a bar with buttons for recalculation control commands,
 - ▼ the list of recalculations as a two-column table: **Status** and **Name**.
- ▼ To set up a custom recalculation, select its name in the Document Tree. The following appears in the right part of the dialog (Fig. 2.10.4):
 - ▼ a bar with buttons for setting recalculation parameters,
 - ▼ the list of recalculated variables as a four-column table:
 - ▼ the **Name** column contains names of variables and/or components,
 - ▼ the **Parameter** column displays the parameter names of the operations whose dimensions need to be recalculated (empty for components),
 - ▼ the **Recalculation method** column is intended for selection of the recalculation method for the value of the variable; each column cell has a drop-down list of recalculation methods: to the middle of the tolerance field, by upper limit, by lower limit, with a coefficient (see Table 2.10.1 on p. 673); if created in the component file, there are custom recalculations in the list for the component; no recalculation with a coefficient is used for components.
 - ▼ The **Coefficient** column serves for setting the coefficient; the coefficient's value can vary from 0 to 1 ($0 < K < 1$).



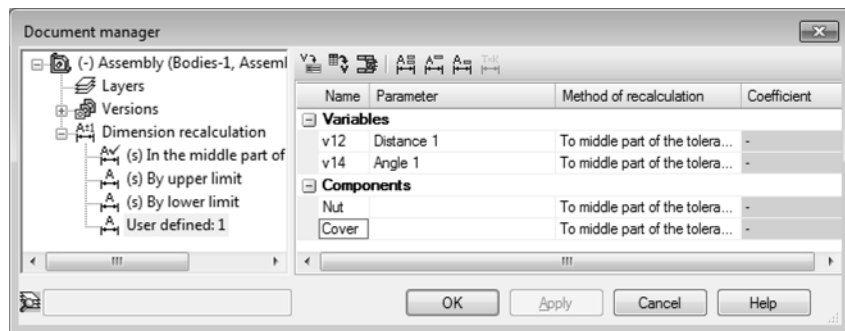
Parameters of a system recalculation are not displayed in the **Document manager** dialog.



a)



b)



c)

Fig. 2.10.4. Display of parameters of a custom recalculation:
 a) the set includes only sketches and operations dimension variables;
 b) the set includes only components;
 c) the set includes both sketches and operations dimension variables and components



The variables, components and recalculation methods in components included in the custom recalculation may be deleted during further editing of the model or components. In this case, the cells related to listed objects are highlighted in red in the **Name** column and cleared in the **Parameter** column.

2.10.3.2.1. Creating, editing, and deleting user recalculations



To create, edit, or delete a custom recalculation, open the **Recalculation control...** menu using the **Recalculate dimensions based on tolerances** button on the **Modes** bar. The **Document manager** dialog will be displayed (Fig. 2.10.3 on p. 674).

In the dialog that appears, perform the following steps.

To close the **Document manager** dialog, click **OK**.



Confirmation of each modification in the **Document manager** dialog is not required, since modifications are saved in the model immediately after their execution.

Creating a user recalculation

In order to create a user recalculation of model dimensions, perform the following actions.

1. Select the **Recalculate dimensions** element in the left part of the **Document manager** dialog.
2. Click the **Create** button on the toolbar of the **Document manager** dialog.
3. Indicate the created recalculation in the list of recalculations in the left part of the dialog.
4. Generate a set of variables and/or components of the new recalculation.



▼ Use one of the following methods to add variables.

▼ In the Design Tree or the model window, specify the derivative dimensions corresponding to the required dimensions of sketches and operations (the derivative dimensions must be created in advance, see Section 2.9.2.8 on p. 648).



▼ Add the desired set of model variables using **Read variables** or **Insert variable** buttons on the toolbar of the **Document manager** dialog (for details, see Section 2.10.3.2.2 on p. 677).



▼ To add a component, select it in the Design Tree or in the model window. Please, note that all first-level components are involved in recalculation of dimensions, i.e. objects or variables of a component cannot be selectively indicated.

Rows corresponding to the variables and/or components of the set appear in the list of variables of the new recalculation.

5. Specify a method to recalculate each variable and component. To do this, select the required method from a drop-down list in each cell of the **Method** column. **to the middle of the tolerance field, by upper limit, by lower limit, using a coefficient** (for more information on the methods of dimensions recalculation see Table 2.10.1 on p. 673). When a method **with a coefficient** is used, enter the value of the coefficient from 0 to 1 in the respective cell of the **Coefficient** column.

Please, note special features of setting up the component recalculation:

- ▼ the method of recalculation **with a coefficient** is not available,
- ▼ when **to the middle of the tolerance field, by upper limit** or **by lower limit** is selected as the method, all component variables are recalculated with this method,
- ▼ custom recalculations created in the component file are available for selection.



To move between cells, use arrow keys.



You can create a new user recalculation using parameters of an already existing system or user recalculation as a template. To create a custom recalculation using an existing one as a template, use the **By sample** button on the toolbar of the **Document manager** dialog (for details, see Section 2.10.3.2.2 on p. 677).

Editing a user recalculation

Editing a user recalculation is very similar to its creation. To start editing, open the **Document manager** dialog by performing the actions described above.

Deleting a user recalculation

A user recalculation may be deleted if it is no longer required. For this, do the following.

1. Open the **Document manager** dialog by performing the actions described above.
2. Select the required calculation 's name in the list.
3. Click **Delete recalculation** on the dialog's toolbar or call the command from the recalculation's context menu.



The selected recalculation is removed from the list.

2.10.3.2.2. Techniques of work with recalculation modes in the Document manager

It is assumed when describing techniques of work with recalculation modes that the **Document manager** dialog is open and **Recalculate dimensions** is selected in the Document Tree in its left part.

Creating a user recalculation



Click **Create** on the toolbar of the **Document manager** dialog or call the command from any recalculation's context menu. A new recalculation is added to the list.

Creating a user recalculation using parameters of an existing recalculation



Select the sample recalculation in the list and click the **By sample** button or call the command from the context menu. The new recalculation with parameters of the sample recalculation is added to the list.

If a system recalculation is selected as a template, all the variables of the current model that correspond to sketches and operations dimensions will be included in the new recalculation. The recalculation method of each variable value in this recalculation will match the one in the selected system recalculation.

If required, you can change the parameters of a recalculation created using a template.

Selecting the current recalculation



Select the necessary recalculation in the list and click the **Make current** button or call the command from the context menu. A check mark appears in the list of recalculations on the icon. This means that the selected recalculation is now active (current), i.e. this recalculation will be used as the basis for recalculation of model dimensions.



There is another method available to make a recalculation current. Double click the necessary line in the list of recalculations to activate it.

Renaming a custom recalculation

Click the necessary cell in the **Name** column to activate it and enter the new name of recalculation. To move between cells, use arrow keys.

Recalculation can also be renamed in the Document Tree. To do this, click the mouse once to select the recalculation name and click again to make it available for editing, then enter the new name.



System recalculations cannot be renamed.

Deleting a user recalculation



Select the necessary custom recalculation in the list and click **Delete** or call the command from the context menu.



Please note that system recalculations cannot be deleted.

Adding a set of variables corresponding to dimensions of graphic documents or operations into a user recalculation



Select the required custom recalculation in the Document Tree and click **Read variables**. All the variables of model operations dimensions and parameters involved in recalculation of its dimensions will be added to the list of variables to the right.

Please note that the variables of the model operations dimensions and parameters that have the status of informational will not be added to the list of variables (see Section 7.1.1.2.1 on p. 1570 for more information about informational variables).

Adding a particular model variable in a user recalculation



Select the required custom recalculation in the Document Tree and click **Insert variable**. The **Variables** dialog allowing to select the required variable from the variables list of the current document appears on the screen. Specify the required variable and click the **OK** button. The chosen variable will be added to the selected recalculation.

Adding components to a custom recalculation

Select the necessary custom recalculation in the Document Tree. Indicate the component to be added by clicking on it in the Design Tree or in the model window. The line with the component's name will be added to the selected recalculation.

Deleting variables and components from a custom recalculation



Select the necessary custom recalculation in the Document Tree. Then select the required lines or any cells of these lines in the right part of the dialog and click **Delete lines**.

Line deletion means that the variable or component corresponding to this line is removed from the custom recalculation. The variable or component remains unmodified in the model.

Selecting the recalculation method of the variable value

Select the necessary custom recalculation in the Document Tree. Then activate the cell of the variable a recalculation method needs to be assigned to in the **Recalculation method** column in the right part of the dialog. Select the desired method of recalculation from the drop-down list of this cell (for information on dimensions recalculation see Table 2.10.1 on p. 673) or click a button corresponding to the desired recalculation method on the toolbar:



▼ To the middle of the tolerance field,



▼ By upper limit,



▼ By lower limit,



▼ Using a factor.

2.10.3.3. Enabling dimensions recalculation mode and working in it



To enable the model dimensions recalculation mode, click **Recalculate dimensions based on tolerances** on the **Modes** bar or call the command of the same name from the **Service — Recalculate dimensions** menu.



The **Recalculate dimensions based on tolerances** button is not available when editing a component in-place.

Menu of the **Recalculate dimensions based on tolerances** command contains the list of all recalculations of the model's dimensions and commands **Recalculation Control...** and **Create Copy**.

This menu is called by clicking on the triangle located near the button.

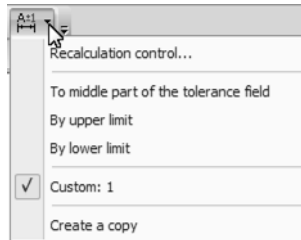


Fig. 2.10.5. The **Recalculate dimensions based on tolerances** button menu contains:



All commands of the **Recalculate dimensions based on tolerances** command menu, except for user recalculations, can also be called from the **Service — Recalculation of Dimensions** menu.

The commands for creation of new objects and for editing existing objects are unavailable if the recalculation mode is enabled. Only the following actions can be performed:

- ▼ model viewing and printing (on printing, see Section 8.1 on p. 1631);
- ▼ measuring geometrical (without creating modification objects) and mass center characteristics of the model (see Section 2.15.2 on p. 812);
- ▼ report creation (see Section 5.2 on p. 1403).

In addition, in the recalculation mode you can:

- ▼ change the current recalculation (see Section 2.10.3.3.1 on p. 680);
- ▼ create a copy of the recalculated model (see Section 2.10.3.3.2 on p. 681).

To exit the dimensions recalculation mode, release **Recalculate dimensions based on tolerances** or click the recalculation mode icon in the document window with the mouse.

2.10.3.3.1. Selecting the current dimensions recalculation

Current recalculation — the recalculation based on which model dimensions are recalculated in the recalculation mode.



The current recalculation is ticked in the **Recalculate dimensions based on tolerances** button menu in the **Modes** bar (Fig. 2.10.5).

To select a current dimensions recalculation, complete the following steps.

1. Open the list of recalculations. To do this, click the triangle located to the right of the **Recalculate dimensions based on tolerances** button
2. Select the desired recalculation from the list.

A checkmark will appear next to the selected recalculation.

Selection of the current recalculation from the **Recalculate dimensions based on tolerances** button menu is available both inside and outside the dimensions recalculation mode. Outside the recalculation mode, the current recalculation can also be selected in the **Document manager** (see Section 2.10.3.2.2 on p. 677).

2.10.3.3.2. Creating a copy of a recalculated model

Working in the model dimensions recalculation mode, you can save the variant of the current model whose nominal dimensions are dimensions recalculated within the current recalculation. By changing recalculation, you can create several model variants with different dimensions — from smallest to largest limits.



Use **Create copy** to save a model with recalculated dimension values. It is in the menu of the **Recalculate dimensions based on tolerances** button in the **Modes** bar (see Fig. 2.10.5) and in the **Service — Recalculate dimensions** menu.



The **Create a copy** command is available in the dimensions recalculation mode only.

After calling the **Create copy** command, the File Save dialog will appear. Use it to specify file name and folder for saving and click the **Save** button.

When you create a copy, the following happens:

- ▼ objects created directly in the model acquire the recalculated dimension values,
- ▼ for model components, the recalculation method that are inserted with changes (see Section 2.16.2.1 on p. 861); component files are not modified on the disk.

A copy of a recalculated model is not different from a model constructed from scratch, i.e. you can work in a usual manner with it in the future.



Unlike **Create copy**, **File — Save as** saves a model without changing nominal dimensions.

2.11. Components

2.11.1. General details about components

Component is part of a model represented by another model. Model components can include parts, assemblies, base and local parts; assemblies can also be standard products and library models.

Component model can be stored in a separate file or current part file.

Separate files are used to store the following components:

- ▼ parts,
- ▼ assemblies,
- ▼ base parts with a history,
- ▼ standard products,
- ▼ library models.

In this case, the current model does not actually contain the components themselves, but rather links to the respective files. This is why it's necessary to transfer component files along with this kind of model in order to move the model to another workplace.

Separate files are used to store the following components:

- ▼ local parts,
- ▼ base parts without production history.

These components do not have special files allocated for them. They are stored in the models that contain them. Using local parts and base ones without production history allows creating a assembly model file, with all the model parts stored in the same file.

To add components to a model the following methods are used:

- ▼ insert component from file (see Section 2.11.2.1 on p. 684),
- ▼ build component in the context of current model (see Section 2.11.2.2 on p. 687),
- ▼ convert current model objects into part/local part (see Section 2.11.2.3 on p. 689).

Components can be linked by matings between themselves or with other model objects. Two types of a matings are available: positioning and mechanical. Positioning matings restrict positions of objects in relation to each other (for example, set the alignment of the pivot and the hole), while mechanical matings set a law of motion for objects in relation to each other (for example, translational movement of the nut during the rotation of the bolt). Creating matings is described in Section 2.11.4 on p. 703.

One and the same model can be inserted as a component into several other models, and, likewise, a model can contain several inserts of a component.

Operations that imitate processing an assembled product can be performed within a component-containing model. Usually these are operations of removing material; for example, making a hole running through several components. Please notice that the result of such an operation is not saved to component files, but is stored in the containing model instead. If you open the component files, you will see that they have no record of the hole. Thanks to this, the same

component can be inserted into different models while performing various operations in them, as well. This will not result in the component file changing on the disk.

If there is a need for changes made to a component to be reflected in all the models that contain it, its file needs to be opened and edited appropriately. A component can be edited either 'in place', i.e. in the screen of the model that contains it (see Section 2.11.6.2 on p. 723), or in a separate screen (see Section 2.11.6.1 on p. 722). For components stored in the current model only on-the-spot editing is available.

If necessary, you can change the position of a component in a model and its insertion settings (see Section 2.11.6.3 on p. 724).

If the components are parts, Boolean operations of subtraction and grouping can be conducted with them (see Section 2.11.5 on p. 720).

By default, the components inserted into a model keep their own mass-center characteristics and properties (designation, title, etc.), i.e. written to their respective files. If necessary, you can set MP (see Section 2.14.6.3 on p. 805) and properties (see Section 5.1.4.1.2 on p. 1387) for a component that differ from its own. This data is not written to the component file, it is stored in the containing model.

2.11.2. Adding a component to a model

Commands to add components are placed in menu **Operations**, and the buttons to call them — on the panel **Assembly Editing**.



While working with a part (*.m3d) the adding commands are available on the **Assembly editing** panel only. To display it select **View — Toolbars — Assembly editing**.

If the model should include several identical components, you can insert them all by copying and pasting the first inserted component the required number of times (see Section 2.14.4.4 on p. 800). If you need to arrange copies of the component in a specific manner (for example, along some curved line, or as a grid with specified parameters), we recommend to use the commands for arrays creation (see Section 2.7 on p. 509).

2.11.2.1. Adding a component from file

Another model (part or assembly) stored in a file on disk can be inserted into a model as a component.

Model insertion from a file can be used for the following components:

- ▼ part,
- ▼ subassembly,
- ▼ local part,
- ▼ base part.

The position of the component in the model is specified during the insertion. The following methods are used for this:

- ▼ **By coordinates**,

▼ By matings.

If a component's position is set using **By coordinates**, and the component is the first one in the current model it gets fixed in position automatically. For details about fixation, see Section 2.11.3.4 on p. 702.

If an inserted component is a model's variant, you can replace it with another variant of the model. For this purpose, call the **Current variant** command from the component's context menu in the Design Tree and specify the required variant in the appearing list of the model's variants. A tick will appear next to the designation of the chosen variant in the list. This will result in inserting the version in the model. After replacing the component rebuild the model (for details about model rebuilding, see Section 2.14.3.7 on p. 795).

Later on, you can change the position of the inserted component within the model, or edit its parameters (see Section 2.11.6.3 on p. 724).

2.11.2.1.1. Adding a part/subassembly from file

To add a part or subassembly stored in a file as a component, perform the following actions.



1. Call the **Operations — Add from file — Component...** command or click the **Add from file** button on the **Assembly Editing** bar.

2. In the open file dialog box, select the file that contains the required model (see Section 2.11.2.5.1 on p. 692).

A ghost of the selected model will appear in the current model screen, and the Properties panel will get two additional tabs: **Position** and **Properties**.

3. Use the **Position** tab to set the component position parameters in the model (see Section 2.11.2.5.2 on p. 692).

4. On the **Properties** tab, set the color and optical properties of the component (control elements used for this are described in Section 2.14.2 on p. 784), turn on/off specification object creation.



On this tab, you can also select new values of model external variables, provided the inserted component contains a variable table. For this, you need to click the **Table of variables** button and select the required row in the table that appears on the screen.



5. To complete component insertion click the **Create object** button on the Advanced Control Panel.



An icon that corresponds to the component type (part or assembly) will appear on the Model Design Tree, while the name of the component retrieved from its source file appears next to the icon.



2.11.2.1.2. Adding local part from file

A part or assembly, stored in a file, can be inserted as a local part.

The model chosen for insertion may contain local parts already. In this case, please take note of the following.

- ▼ If the chosen model is an assembly, then either the whole of the assembly or any one of its local parts can be inserted.

- ▼ If the chosen model is a part, only the whole of the said part can be inserted. No local part it may contain will be available for selection.

To add a local part to a model, perform the following actions.



1. Call the command **Operations — Add from file — Local part...** or click the **Add local part from file** button on the **Assembly editing** panel.

2. In the open file dialog box, select the file that contains the required model (see Section 2.11.2.5.1 on p. 692).

If the chosen model is an assembly that in turn contains local parts a **Select local part** dialog box will appear on the screen.

To insert one of the local parts in an assembly as a local part, specify it in the dialog box and click **OK**.

To insert the whole of the assembly as a local part, click **Cancel**.

After selecting the source model a ghost of the local part will appear in the current model screen, while the Properties panel will get additional two tabs—**Position** and **Properties**.

3. On the **Position** tab set the position parameters for the local part within the model (see Section 2.11.2.5.2 on p. 692).
4. On the **Properties** tab, set the color and optical properties of the component (control elements used for this are described in Section 2.14.2 on p. 784), turn on/off specification object creation.



On this tab, you can also select new values of local part external variables, provided the source model contains a table of variables. For this, you need to click the **Table of variables** button and select the required row in the table that appears on the screen.



5. To complete local part insertion click **Create object** button on the Advanced Control Panel.

Local part will now be inserted in the current model.



An icon that corresponds to the component type (part or assembly) will appear on the Model Design Tree, while the name of the component retrieved from its source file appears next to the icon.

The local part is stored in the file of the model it has been inserted into.

2.11.2.1.3. Adding Base part

A convenient technique in modeling products that differ only in few design details is to use a previously created model (part or assembly) as a base part.

The model that you plan to use must be written to the disk in a file named however you like.

To insert a finished model as a base part, perform the following actions.



1. Call the command **Operations — Base part...** or click the **Base part** button on the **Model editing** panel.

2. In the open file dialog box, select the file that contains the required model (see Section 2.11.2.5.1 on p. 692).

A ghost of the selected model will appear in the current model screen, and the Properties panel will get two additional tabs: **Position** and **Properties**.

3. On the **Location** tab, proceed as follows.

- ▼ Set the base part's position parameters in the model (see Section 2.11.2.5.2 on p. 692).
- ▼ Select base part insertion method. To do this, activate the applicable switch in the **Insertion method** group.



If the **Insertion by external link** switch is enabled, the link between the base part and the source file is kept. The base part is not copied into the model, but exists in it as a link to its file. If the source file is subsequently edited, the changes will be automatically sent to the model that uses this base part (model rebuilding is required for transmitting changes, see Section 2.14.3.7 on p. 795).



If the **Insertion without history** switch is enabled, the link between the base part and the source file is not kept. Editing the source file does not affect the base part inserted using this method.

- ▼ If the base part is required to be a mirror image of a model from a file, turn the **Mirrored insertion** option on. With this option on the base parts will be positioned symmetrically to their initial position relative to the selected plane. The plane is selected from the **Symmetry plane** list, which becomes available after the option is turned on.

4. On the **Properties** tab, set the base part's color and optical properties. Control elements used for this are described in Section 2.14.2 on p. 784.



On this tab, you can also select new values of the base part's external variables, provided the source model contains a variable table. For this, you need to click the **Table of Variables** button and select the required row in the table that appears on the screen.



5. To complete base part insertion click **Create object**.

The base part will now be inserted in the current model. An icon that corresponds to the base part insertion method used will appear on the Model Design Tree:



- ▼ base part inserted using an external link,



- ▼ base part without history.

The name of the base part retrieved from the source file will appear next to the icon.

If the position of the base part is set using the **By coordinates** method, it is fixed automatically.

2.11.2.2. Creating component 'in place'

While forming an assembly model in KOMPAS-3D, besides adding ready-made components from the disk, you can also create them without leaving the current file, i.e. construct parts and subassemblies in the context of the current model. All the remaining components will be visible in the window, too. They are unavailable for editing, but their elements (facets, edges, summits, outlines, etc.) can be used in the operations of creating new components.

The component (part, subassembly, local part) created in the context of the current model is linked with the model by a *Coincidence* matings (see Section 2.11.4.2.2 on p. 707). The matings is referenced to the component's absolute coordinate system and the model's relative coordinate system. The mating's icon appears in the group of matings of the Design Tree. If necessary, you can edit or delete the automatically-created mating.

Later on, you can change the position of the created component within the model, or edit its parameters (see Section 2.11.6.3 on p. 724).



When creating a component within a model you need to name a file for the component and select a folder to save it in. If a file with the same name already exists in the selected folder, it is not possible to overwrite it. You need to change the name of the file or select a different folder to save it in. In addition, if a model from this file is a component of one of the open models, attempting to overwrite results in interrupted creation of the component.

2.11.2.2.1. Creating a part 'in place'



To start building a part directly within the current model, call the command **Operations — Create — Part** or click the **Create part** button on the **Assembly editing** panel.

In the open file dialog box that pops up specify the folder and the name of the file to save the new part to and click **Save**.



An icon designating the new will appear on the Model Design Tree. The system will automatically switch to context editing mode for the part. Operation in this mode is described in Section 2.11.6.2.1 on p. 724.



As you finish building the part, release the **Edit in place** button on the **Current status** panel, and use the context menu to call the command **Edit in place**, or click the context editing mode icon in the model screen. The system will return to main model operation mode.



If you plan to build the first element of the part being created based on an outline, and a 2D object to place it on is already available in the model, specify the object before calling the part creation command. In this event, after saving the file of the part and turning on the context editing mode, the system automatically switches to editing mode for the outline located in the specified plane. Make required constructions and exit the sketch mode.

2.11.2.2.2. Creating subassembly 'in place'



To begin building a subassembly directly in the current model, call the command **Operations — Create — Assembly** or click the **Create assembly** button on the **Assembly editing** panel.

In the save file dialog box specify the folder and the name of the file to save the new assembly to, and click **Save**.



An icon designating the new subassembly will appear on the Model Design Tree. The system will automatically switch to the building mode for the subassembly in the context of the model that contains it. Operation in this mode is described in Section 2.11.6.2.1 on p. 724.



As you finish creating a subassembly, release the **Edit in place** button on the **Current status** panel, or use the context menu to call the command **Edit in place**, or click the context editing mode icon in the model screen. The system will return to main model operation mode.

2.11.2.2.3. Creating local part 'in place'



To start building a part directly within the current model, call the command **Operations — Create — Local part** or click the **Create local part** button on the **Edit assembly** panel.

Creating a local part 'in place' is similar to creating a part 'in place' (see Section 2.11.2.2.1 on p. 688) with only one difference: it is not required to enter the name of the file of the part since a local part is stored in the same model where it is created.

2.11.2.3. Transforming objects

To create model components you can use transforming objects available in a given model into a part. While working on an assembly objects can be transformed into a local part.

Transformation can only be performed on objects built directly in the current model. These include:

- ▼ bodies,
- ▼ surfaces,
- ▼ 3D curves and points,
- ▼ local coordinate systems,
- ▼ auxiliary planes,
- ▼ auxiliary axes.

If necessary, you can transform objects built within a component into a part. This requires entering the editing mode for the component.



Transforming objects into a part or local part cannot be undone. Nor is it possible to undo operations performed within the model before the transformation.

The following transformations are available for components: transforming a part into a local part and, vice versa, a local part into a regular part.

- ▼ To transform a usual part into a local one, select it on the design tree and call the command **Embed into document**. While working with an assembly the command is called from the **Editor** menu or a context menu; while working with a part, it can only be called from the context menu. After calling the command, the selected part is removed from the model and re-inserted into it as a local one.
- ▼ To transform a local part into usual part, select it on the Design Tree and call the **Save as...** command from a context menu. The Save file wizard dialog will appear. Set the name of the file and specify the folder to save it in. Once it is saved, the local part is removed from the model and re-inserted into it as usual one.

2.11.2.3.1. Transforming objects into a part

Transforming objects into a part is to save the objects built within a model (part or assembly) to the file of a new part (*.m3d). If a transformation is conducted within an assembly, the resulting part is inserted into it as a component.

While working with a part, objects selected for transformation are transmitted to the new part and saved in the original one unchanged.

While working with an assembly, objects selected for transformation can be deleted from the assembly in the process of transformation.

Transforming objects into a part is performed using the command **Editor — Create part**.

You can select objects you need both before and after calling the command. The objects to be transformed are specified on the Design Tree or model screen (to specify a body or surface you can select its any facet, rib or summit).



If you have selected a body (or surface) on the Design Tree, you can call the **Create part** command from the context menu.

Upon calling the command, the transformation controls appear on the Properties bar. To set transformation parameters use the following steps.

1. Set the designation of the part being created. To do so, click the **Designation** field and follow the instructions in the dialog that opens. Working with the dialog is described in Section 2.13.2.1.1 on p. 756.
2. In the **Name** field, enter the name of the part being created.
3. For a set of objects being transformed using the **List of objects** panel. Objects selected prior to calling the command are already included in the list. To add an object to the list specify it on the Design Tree or model screen (to specify a body or surface in the model screen, you can select any of its faces, edges, or vertices). Specifying the object a second time will result in deleting it from the list. You can also delete an object by selecting it on the list and clicking the **Delete from list** button.



You can add only those objects to the list whose types are listed in see Section 2.11.2.3 on p. 689.



4. Using the **Method** group of switches select a method to save the objects to the file of the part. **Saving with history** option button is active in this group by default. This means that the new part will receive both the objects themselves and those of the original objects whose transmission is possible. In the event it is impossible to transfer the original objects the following data is transferred:
 - ▼ if transformation takes place in a part, copies of the geometry of original objects will be created in the new part (for details about copying geometric objects, see Section 2.8.7 on p. 624),
 - ▼ if transformation takes place in an assembly, links to the assembly file will be generated in the new part, similar to those that are formed in the component file during its editing 'in place' provided objects from other assembly components are used.



A Boolean operation on a body is transmitted to the part correctly only when the other body taking part in the operation belongs to the current model itself, not to one of its components.



A part obtained from a body produced by a *Add thickness* returns an 'Cannot perform this operation' error. If the operation modifies the body instead of producing it, the transformation will be successful.

When saving with history, apart from initial objects, all expressions used for calculating parameters of elements of the objects being transformed will be passed to the part, as well as all main section's variables.

The position of the objects is saved during the transmission, i.e. the transmitted objects will be located in the part's coordinate system exactly the same way as in that of the original model.



If you need to pass only objects themselves to the part, without initial objects and variables, turn on the **Save without history** switch in the **Method** group. The **Mirroring** option will become available, too. Enable this option if it is required that all objects passed to the part be positioned symmetrically to their initial position relative to one of the coordinate planes. The appropriate plane is selected from the **Symmetry plane** list, which becomes available on enabling the option.

5. Additional operations are available when working with assemblies.
 - ▼ Enable the **Composition geometry** option, if you need to add a new part to the assembly not as a component, but as a composition geometry.
 - ▼ Enable the **Delete original** option, if you want the objects to be deleted from the assembly after transformation. Please note that only objects selected for transformation can be deleted. Their initial objects remain intact.



If the object transformed into the part was involved in mating, then, in case of deleting the original, a new part will participate in relevant matings; if the original is saved the matings will not be changed.



6. After setting all transformation parameters, click the **Create object** button on the Advanced Control Panel. Windows-standard File Save dialog will be displayed. Specify both folder and part file name for saving, then click the **Save** button. The selected objects will be saved to the specified file.

2.11.2.3.2. Transforming objects into a local part

Transforming objects into a local part is possible only when working with assemblies.

Objects built in a current assembly are transformed into a local part of the same assembly. The created local part is automatically added to the list of the current assembly's components.

Transforming objects into a local part is performed using the command **Editor — Create local part**.

Transforming objects into a local part is similar to transforming objects into a part (see Section 2.11.2.3.1). The difference is that transforming into a local part does not create a separate part file (*.m3d). Local part is stored in the file of the same assembly in which it is created.

2.11.2.4. Adding standard item

If standard items (bolts, nuts, screws, etc.) are used in a product, you do not need to model them as unique parts. Standard item models can be inserted from **Standard item library**.

If **Standard item library** is installed on your workstation, the **Libraries** menu will have the **Standard items** item. It contains a submenu of commands for access to library functions.

To insert a standard item into a model, call the command **Libraries — Standard items — Insert — Insert element**. In the appearing dialog, double-click the required standardized part, set its parameters, and click **Apply**. Indicate the part's insertion point in the model window or objects for mating and then click the **Create object** button on the Special control bar.

The inserted item shall appear in the model window and its icon — in Design Tree.

More information on the **Standardized parts library** can be found in its help system.

Main methods of working with standardized parts — shift and rotation, matings, setting properties, access management and others — are the same as when working with a unique component (part, subassembly).

2.11.2.5. Position and component insertion parameters

When adding a component to a model from a file, there are several methods to set its position in the model (see Section 2.11.2.5.2 on p. 692). Position control elements are found on the **Location** tab on the Properties panel.

In addition, you can use the **Properties** tab to set the component's properties, select new values of external variables using the source model's variable tables.

2.11.2.5.1. Selecting component file

After calling one of the commands for inserting a component into the model, an insert source file selection dialog box will appear.

Specify the required model file.

If the model contains variants, geometry and properties of the component will correspond to one of these variants. The list of all variants, that exist in the model, is contained in the **Variant** list of the file selection dialog. Specify the required variant in the list.

If additional numbers of variants are created in the chosen model, the dialog will also contain the **Additional Number** list. By default, the list has the activated variant under which none of the additional numbers is selected (empty line of the list). If necessary, choose the line with the required number.

To complete selection of the model, click **Open**.

2.11.2.5.2. Setting component position parameters

The parameters that determine the position of the component are set on the **Location** tab on the Properties panel.



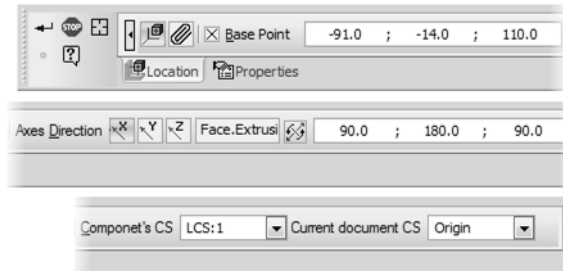
Setting parameters depends on the method of component positioning: **By coordinates** or **By matings**. You can select the method you want using the group of **Location method** switches.



The set of elements and component positioning procedure for each method are given below.

By coordinates location method

Control elements displayed on the **Location** tab while using the **By coordinates** method are shown in Fig. 2.11.1.

Fig. 2.11.1. **By coordinates** method

To set the position of a component using this method, perform the following actions.

1. Set the position of the insertion point for the components, i.e. the same location as the origin of the current coordinate system of the component.

The following ways to set the position are available.

- ▼ Manual in the model screen. The insertion point can be specified at lib or using a snap (for example, to the coordinate origin or to a vertex). To make the snap, position the cursor on the required object and left-click the object's symbol as soon as it appears next to it.



Snap to the zero point of the current coordinate system can be made using the keyboard. For this purpose, press **<Ctrl>+<O>** and then the **<Enter>** key.

- ▼ Inputting coordinates in the **Base point** field.



- ▼ Point creation. To do so, click the **Create Point** on the Advanced Control Panel. This will initiate the process of point creating (see Section 2.5.2.11.1 on p. 347). Select a construction method and specify the required parameters. These parameters will determine the position of the component's insertion point. Please note that in this scenario the point construction process is used solely for determining the position of the insertion point. No new point is created in the process.

The position of the insertion point is fixed using an option next to the **Base point** field. Cross corresponds to the on state of the option (i.e. the insertion point is fixed), while a check mark indicates that it is in the off state.

The option turns on automatically in the following cases:

- ▼ after setting the insertion point manually in the model screen using the above methods,
- ▼ after inputting all three coordinates in the **Base point** field,
- ▼ upon completion of the point creation process.

When needed, you can turn the option on/off manually.



Editing coordinates in the **Base point** field and creating a point using the **Create Point** command are available regardless of whether the option is enabled or not.

You can manually set the insertion point even if the option is off. If the option is on, you can only drag the component beyond the origin of its coordinate system.

2. Set the orientation of the component's coordinate system using guiding objects or Euler angles.

To select the axis to set the respective direction, enable the appropriate switch in the **Axes Direction** group:



- ▼ **Guiding object for X axis,**



- ▼ **Guiding object for Y axis,**



- ▼ **Guiding object for Z axis.**

The guiding object display field and a button for changing the axis direction to the opposite will appear on the Properties panel. The button corresponds to the selected axis:



- ▼ **Flip the X axis,**



- ▼ **Flip the Y axis,**



- ▼ **Flip the Z axis.**

Specify a guiding object in the model screen. The name of the object will appear in the **Axis guiding object** field. The axis will establish an associative link with this object.

If necessary, flip the axis direction using one of the button described above.

Direction can be set for two axes only. The third axis will be automatically directed in such a way so as to form a right-hand system with the first two.

After you select a guide object for the first axis, the component coordinate system will rotate so that this axis has the direction specified by this object.



The guide object direction for the second axis can be non-perpendicular to the first axis. In this case the direction of the second axis is defined as a projection of the direction, defined by the object, on the plane perpendicular to the first axis.

If the object is either parallel to or on the same axis with the first axis, such object can not be used as a guide for the second axis.

To set axis orientation you can use the **Euler angles** field. The field is used to input nutation, precession and rotation angles. In case you select guiding objects, these angles are calculated automatically and their values appear in the field. You can change the value of the angles at any time. As you do so, the current selection of guiding objects will be canceled.



You can control the position of the component in the model using a base element (see Section 2.11.2.5.3 on p. 696).

3. You can control the position of the component in the model using coordinate systems. Initially, the component is placed in a model as follows:
 - ▼ the origin of the component coordinate system coincides with the insertion point,
 - ▼ axes of the component's current coordinate system are co-directional with those of the model's current coordinate system,
 - ▼ coordinates of the insertion point and Euler angles are determined relative to the current coordinate system of the model.

You can change the position of the component in the model by specifying another coordinate system or current model. Use the respective lists for this purpose. Availability of lists depends on the availability of local coordinate systems in the component/current model.

The component's coordinate system is chosen from the **Component CS** list. It is combined with the insertion point and location of the component changes.

In order to choose a coordinate system for the current model, use the **Current document's CS** list. If the CS is chosen while the **Save position** option is disabled, the coordinates of the insertion point remain unchanged but are counted in the chosen coordinate system. As a result, the component's position will be changed in the model. If the **Save position** option is enabled, the choice of the model's CS does not affect the component's position in it. The coordinates of the insertion point and Euler angles change (their values are determined relative to the current coordinate system).



If necessary, you can change the position of the component's coordinate system when placing the component. For this purpose, click the **Change component's CS position** button in the Advanced Control Panel. The LCS creation process will start (see Sections 2.8.3.6 on p. 602 and 2.8.3.7 on p. 606). Specify LCS parameters and click the **Create object** button. The system will return to the component construction process. In the **Component CS** list, the *Modified* line will appear. Please note that in this scenario the LCS creation process is used solely for determining the position of the component's CS. No new LCS is created in the process.



Note the following feature when setting a new position for CS. If the **Create LCS only in absolute CS** option is enabled during the system setup in the **LCS** dialog (see Section 9.1.11.9 on p. 1734), the CS position is defined relative to the component's absolute CS; otherwise, it is defined relative to the current CS.



CS position change and selection of another CS are only used in the current component location process. This data is not saved in the system.

Matings positioning method

Control elements displayed on the **Location** tab when the **By matings** method is selected are shown in Fig. 2.11.2.

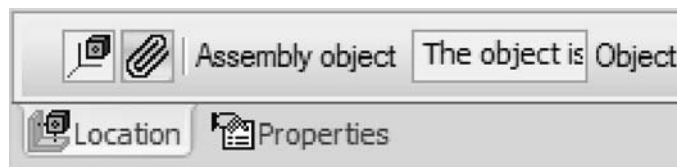


Fig. 2.11.2. **By matings** method

In addition, as you select this method, the source model screen appears in the model screen. It is the same screen as one for copying objects from an external file (see Section 2.8.7.1.1 on p. 627).



When inserting a model from the application, the source model screen is not displayed.

To set the position of a component using the **By matings** method take these steps.

1. Specify the mating objects of the current model and component on the Design Tree or in the model screen (component object can be specified in the source model screen). The names of the objects will appear in the **Assembly object** and **Component object** fields, respectively.



If you need to cancel object selection, specify it again. The **Select other** button makes it possible to cancel the selection of all objects.

2. Select a type of positioning matings by turning on the respective switch in the **Mating selection** group (the switches available depend on which objects have been selected for mating). The process of creating a matings of the set type will commence. The techniques for setting up positioning matings of different kinds are discussed in Section 2.11.4.2 on p. 706.



You can select objects for mating in the middle of the process of creating a matings.

Upon the completion of matings creation the system will return to the initial process. The created matings will be displayed on the **Matings** panel. The panel includes a matings list and list control elements.

Create the remaining matings using the same procedure.

You can include matings in the calculations or exclude them, by turning on or off respective options on the list.



To change the parameters of a matings, select it on the list and click the **Edit** button. The matings editing process will commence. Make necessary changes and click the **Create object** button on the Advanced Control Panel.



To delete a matings, select it on the list and click **Delete**.

2.11.2.5.3. Component base element

You can change the position of the component in the model using a special tool: **base element**.

The base element coincides with the component's current coordinate system. The element consists of three 3D arrows of red, green and blue corresponding to the positive directions of the X, Y, Z axes of this coordinate system. Arcs of respective colors are shown on the planes that are perpendicular to the arrows. For example, the axis X arrow is red. The arc on the plane that is perpendicular to the arrow is also red. The central point of the element is given as a sphere. An overview of a base element is shown in Fig. 2.11.3.

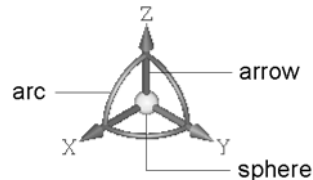


Fig. 2.11.3. Base element

To activate the base element select its arrow or arc. You can select an arrow when an axis icon appear next to the pointer. The rotation icon next to the pointer indicates that you can select an arc.

You can modify the base element's size. To do so, hover the cursor over it and press `<Alt> + <Ctrl> + <Shift>`. Then, keeping the keys pressed, rotate the mouse wheel. The resulting size of the base element will be saved in the system.

You can change a component's position in the model by moving and/or rotating the base element.

▼ Base element movement

- ▼ Along the axis any distance.
Move the cursor to the respective arrow, hold down the left button of the mouse and move the pointer along the selected axis in the appropriate direction. When the base element is in the correct position, release the button.
- ▼ Along the axis at any distance.
Click the arrow that correspond to the required axis. A distance input field will appear on the screen. Type the required value and press `<Enter>`. The base element will move the set distance along the selected axis.
- ▼ Moving in any direction.
Move the cursor to the sphere in the center of element, hold down the left button of the mouse and move the pointer in the required direction. When the base element is in the correct position, release the button.

▼ Base element rotation

- ▼ Around the axis to any angle.
Move the cursor to the arc on the plane that is perpendicular to the required axis. Hold down the left button of the mouse and rotate the coordinate system around the selected axis in the required direction. When the base element is in the correct position, release the button.
- ▼ Around the axis to a predefined angle.
Click the arc on the plane that is perpendicular to the required axis. A angle input field will appear on the screen. Type the required value and press `<Enter>`. The base element will rotate for the set angle around the selected axis.
- ▼ Around the axis with a set step.
Hold down `<Ctrl>` and click the arc on the plane that is perpendicular to the required axis. The base element will rotate one preset step.
If you need to rotate the base element a preset step in the opposite direction, click the arc while holding down `<Ctrl> + <Shift>`.



The step is set in preferences (see Section 9.1.11.5 on p. 1728). By default, it is 15°.

- ▼ Rotating the axis 180°.
 - Click the arrow that correspond to the required axis while holding down **<Ctrl>** or **<Ctrl> + <Shift>**. The axis will rotate 180° around one of two other axes of the base element.
 - While holding down **<Ctrl>**:
 - ▼ X axis around Z axis,
 - ▼ Y axis around X axis,
 - ▼ Z axis around Y axis.
 - While holding down **<Ctrl> + <Shift>**:
 - ▼ X axis around Y axis,
 - ▼ Y axis around Z axis,
 - ▼ Z axis around X axis.

2.11.3. Moving and rotating components

When inserting a component or after inserting it, you can use special commands to set the component's position and orientation in the model, as well as define its position with respect to other components.

KOMPAS-3D has a few methods of moving components within the coordinate system of the model that contains them. You can rotate a component about the center of its outline parallel-epiped, about an axis or about a point, as well as move it in any direction. In order to perform any other rotations of the component, the **Rotate component** command is used. The **Move component** command is used to move the component. The commands can be found in the **Service** menu, and the buttons to call them — on the **Assembly editing** panel.



When working with a part (*.m3d), component movement commands are only available on the **Assembly editing** panel. To display it, use the following command **View — Toolbars — Assembly editing**.

Note the following features when moving a component.

- ▼ A fixed component (see Section 2.11.3.4 on p. 702) cannot be moved in the coordinate system of the model that contains it. In the Design Tree, a fixed component is marked with letter **f** in parentheses.
- ▼ A component's movement may be restricted or prohibited by mates applied to this component (for more information about mates, see Section 2.11.4 on p. 703).
 - ▼ If the component can be moved (such as coaxial components that can be moved along their common axis and rotated about it), it shall be marked with a minus (“-”) sign in parentheses in the Design Tree.

- ▼ If the component cannot be moved (such as a component, coordinate planes of which correspond to coordinate planes of the assembly, so it cannot be moved or rotated), it shall be marked with a plus sign in parentheses in the Design Tree.

After any movement of a component its icon on the Design Tree is marked with a red check mark. This is an indication that the model needs to be rebuilt. To rebuild the model, call the command **View — Rebuild**. For more details about rebuilding models, see Section 2.14.3.7 on p. 795.



One should differentiate between commands for moving components within the model's coordinate system and those for moving the whole model within the screen (see Section 2.1.3 on p. 117).

2.11.3.1. Component Move



To shift a component, call the command **Move component**.

The shape of the cursor will change.



Place the cursor over the component to be moved, click the left mouse button and, while holding it, move the cursor. The component shall move in the same direction. When the desired component position is reached, release the mouse button.



The **Modes** group of switches makes it possible to perform the following:



- ▼ **Simple movement** — movement without additional operations,
- ▼ **Auto matings** — movement with automatic application of matings (see Section 2.11.3.3.1 on p. 700),



- ▼ **Detect Collisions** — movement with detection of components' collisions (see Section 2.11.3.3.2 on p. 700).



To exit from any command of component shift, press the <Esc> key or the **Interrupt command** button on the Advanced Control Panel.

2.11.3.2. Component Rotation



In order to rotate a component, call the **Rotate component** command.

The shape of the cursor will change.



You can rotate a component as follows:

- ▼ around the center of the outline parallelepiped of the component,
- ▼ around a point — a vertex, the origin of a coordinate system, or a point in the sketch,
- ▼ around a straight object — a construction axis, edge, or a line in the sketch.

By default, the component is rotated around the center of the outline parallelepiped.



In order to rotate a component around a point or straight object, activate the **Center/Axis of revolution** switch and specify the required object in the Design Tree or in the model window.

Place the cursor over the component to be rotated, click the left mouse button in the model window and, while holding it, drag the cursor. The component will rotate about the selected element.



The **Modes** group of switches makes it possible to perform the following:



- ▼ **Simple rotation** — rotation without additional operations,



- ▼ **Auto matings** — rotation with automatic application of matings (see Section 2.11.3.3.1 on p. 700),

- ▼ **Detect Collisions** — rotation with detection of components' collisions (see Section 2.11.3.3.2 on p. 700).



To exit from any command of component rotation, press the <Esc> key or the **Interrupt command** button on the Advanced Control Panel.

2.11.3.3. Additional component movement options

When moving a component using any method (shift or rotation), the following options are available:

- ▼ automatic application of matings to the elements of components,
- ▼ detection of components' collisions.

These options are only available for components that are fully loaded to the model, i.e. having the **Full** load mode (see Section 2.12.2.1 on p. 731).

2.11.3.3.1. Automatic Mating in the Process Displacement

When shifting or rotating an assembly component, you can use the automatic mating mode. This mode allows to identify the approaching elements on the components moved towards each other (such as faces, vertices, edges) and automatically add mates corresponding to their shape and type.

The mode of automatic mating is enabled after calling the command for displacing the component.



In order to enable the mode to automatically apply matings, go to the **Modes** group of the Properties bar and activate the **Auto matings** switch.

Move the component. When it approaches the other component, those faces, which can be automatically mated, will be highlighted. If you release the button of the mouse at the moment, when faces are highlighted, then they will be mated.

For example, when two flat faces approach each other, the system on the fly applies a *Coincident* mate to them, while upon cylindrical faces approaching each other — the *Coaxial* mate.

2.11.3.3.2. Detect Collisions

When shifting or rotating an assembly component, you can use the collision detection mode. In this mode, components displacement is restricted by their shape and dimensions: the motion is possible only up to “contact” with another component.

The mode of collision detection is enabled and can be customized after calling the command to displace a component.



In order to enable the collision detection mode, activate the **Detect Collisions** switch in the **Modes** group of the Properties bar. Switches for mode setup will appear in the Properties bar.

Selection of displaced components for collision detection

Movement of a component is usually accompanied with the movement of its mated components.



To perform the collision detection only for the component being displaced, activate the switch **Displaced Component Only** in the group **Detect Collisions**.



To make the collision detection performed for any of all concurrently displaced components, activate the **All components** switch.



If the displaced component is not involved in mates, then the state of switches in the group **Detect Collisions** is of no importance.

Face Highlighting at Collision



To make the contacted faces highlighted in the model window in case of collision of the displaced component with another component in the assembly, activate switch **Enable highlighting faces at collision**.



To disable highlighting of the touching faces, activate the switch **Disable highlighting faces at collision**.

Sound Alert at Collision



To enable a sound alert when one moving component collides with another component, activate the switch **Sound alert at collision on**.



To disable a sound alert, activate the switch **Sound alert at collision off**.

Stop at collision



To make further movement in the current direction impossible if a moving component collides with another component, activate the switch **Stop at collision**. In other words, this option allows you to prevent possible intrusion of the displaced component into other components.



To allow the component to be moved after collision, activate the switch **Do not stop at collision**.

Selection of immovable components for collision detection

By default, the detection of collisions of the components being moved with all other components of the model is enabled.



You can select the particular components to detect collisions for. To do this, activate the switch **Components** and select the required components. Their names will appear in the reference table **List of Components**.



To exclude the component from the list, reselect it or highlight it in the list and click the **Delete** button or press the *<Delete>* key.

2.11.3.4. Fixation of a Component

Fixing a component makes impossible any movement of the component in the coordinate system of the model that contains it. It is recommended to fix at least one component of a model, in order to have predictable movement behavior of components when applying matings.

The first component inserted into a new model from a file and positioned using the **By coordinates** method (see Section 2.11.2.5.2 on p. 692) is fixed automatically.

To fix a component(s) in its current position:

1. Select the component(s) in the Design Tree or in the Model window.

For more details about selecting components, see Section 2.1.4.1 on p. 130.



2. Click the **Enable fixation** button on the **Edit Assembly** bar or call the respective command from the context menu.

Fixed component icons are marked in the Design Tree with the letter **f** in parentheses.

When fixing a couple of components involved in the mate, the mate will automatically be excluded from the calculation. If only one of the mated components is fixed, the mate will not be excluded from calculation.

If necessary, the mate which is automatically excluded from calculation can be included in calculation manually.

If the mate participant component's (components') fixation is disabled, the matings previously excluded from calculations will be included into the calculations again. Mating of components excluded from calculation manually will not be included in calculation automatically.

In order to disable component(s) fixation:

1. Select the component(s) in the Design Tree or in the Model window.



- Click the **Disable fixation** button on the **Edit assembly** panel or call the respective command from the context menu.



The **Enable fixation** and **Disable fixation** commands are not available for components that are array's instances.

Since the property of being fixed is one of component properties, you can use the following method to fix an individual component.

1. Select the component(s) in the Design Tree.
2. From the context menu call the **Properties command**.



In group **Fixation** on the Property bar there will be active switch **Do Not Fix Component**.



3. Activate the other switch from this group — **Fix Component**.



4. Confirm changing of the component properties by clicking the button **Create Object**.

To disable fixation of a particular component, you can use switch **Do Not Fix Component** on the Property Bar.



Note that fixation can be enabled and disabled only for components of the first level of the model being modified. Thus, when working with an assembly, for instance, you can enable or disable fixation for a component contained by this assembly, not by its subassemblies. Likewise, when modifying a subassembly 'in place', you can enable or disable fixation for a component contained by this subassembly. You cannot do the same for a component contained by another subassembly or the assembly.

2.11.4. Mating components

Once all required objects have been created in the model, you can proceed with creating matings.

Matings is a relation between assembly components and objects.

For example, after application of the *Parallelity* mate on faces of two different components, these components become mated. Position of components shall change so that these faces become parallel. Herein after, once any of mated components is moved, the second component moves automatically in order to maintain parallelism of faces.

Various objects can be mated, both belonging to components, and the model that contains them: coordinate planes and axes, coordinate origins, facets, edges, vertices of bodies and surfaces, points, vertices of curves, segments of polylines, graphics objects in sketches, as well as construction axes and surfaces and local coordinate systems.

KOMPAS-3D supports mates of the following types:

- ▼ **Positioning matings** (fixing an object with respect to another object in a certain way):
 - ▼ Coincident,
 - ▼ Tangency,
 - ▼ Coaxial,
 - ▼ Parallel,
 - ▼ Perpendicular,
 - ▼ Placing elements at specified distance,
 - ▼ Placing Elements at Specified Angle;
- ▼ **Mechanical matings** (determines the rule of moving an object with respect to another object when one of them is moved):
 - ▼ Rotation– rotation,
 - ▼ Rotation– move,
 - ▼ Cam– pusher.

Commands to apply positioning mates are available in the menu **Operations— Component Mates**, mechanical mates— in the menu **Operations— Mechanical Component Mates**. Buttons to call these commands are on panel **Mates** (Fig. 2.11.4).

Fig. 2.11.4. Panel **Mates**

When working with a part (*.m3d) component mating commands are available on the **Matings** panel only. To display it, use the command **View – Toolbox – Matings**.

2.11.4.1. General Information about Mates

Positioning matings are, typically, applied during product setup. For more information about creation of positioning mates see Sections 2.11.4.2.1–2.11.4.2.8 on p. 710.

Positioning matings can be applied automatically while moving components. For more information about automatic mating see Section 2.11.3.3.1 on p. 700.

Mechanical mates are auxiliary mates applied for preliminary evaluations and visualization of mechanism operations. For more information about mechanical mates see Sections 2.11.4.3.2–2.11.4.3.5.

Both mate types can be independently applied, edited or deleted. Viewing mechanical mates can be restricted by some existing positioning mates. Therefore, mechanical mating is recommended after the positioning ones.

During creation of matings in the model, variables corresponding to these matings' parameters are created automatically. For more details about creating and using variables, see Section 7.1 on p. 1569.

2.11.4.1.1. Mate Objects

Matings are applied to paired objects, one of which is a component. The same objects may participate in several mates; it is also possible to apply multiple mates onto the same pair of objects.

Initially — right after having been inserted into a model— a component can move freely in its coordinate system.

As a result of applying a positioning mate, a component's number of degrees of freedom is decreased. For example, if you set coincidence of a part face with a plane, then this part shall only have three degrees of freedom: two degrees of freedom - for displacement, and one - for rotation.



The first component inserted into a model from a file is fixed automatically. If required, you may fix any other components, or cancel any component fixation (see Section 2.11.3.4 on p. 702).

After all positioning matings are applied, it is recommended to fix assembly components in the model coordinate system.



If in your future work you plan to apply mechanical mates, the positioning mates shall be applied in such a way that the objects could have the required degrees of freedom.

The Design Tree uses the following symbols which indicate whether a component may move in the model coordinate system:

- (f) — fixed component,
- (+) — a component with fully defined positioning matings, i.e. having no freedom degree in the model coordinate system,
- (-) — not fully defined component.

A definition is added before a component's name.



The root object of the Tree, the *Components* section, its group or a branch of identical objects have the (+) sign in front of their names, if all objects included in them are fixed or fully defined. In other cases, they have the (-) sign.

Besides fixed and fully defined components, the following objects also remain fixed in the model coordinate system:

- ▼ objects belonging to the model on the whole, not one of its components (these objects are found on the first level on the Design Tree)
- ▼ array instance.



Instances of the array being, in turn, the instance of another array cannot be involved in mates.

If one of two mated components is fixed, then the other component mobility (and, therefore, it's further mating ability with other objects) is more limited, as opposed to it's mating with a "free" component.

It is impossible to mate two fixed objects (in some cases, the mate is created and marked immediately as "error"). For example, you cannot set coincidence of two axes, which are assembly elements, even though they pass through edges or vertices of different parts. It is also impossible to mate objects that belong to the same component: this requires objects in the component to be independently moved, while this component moves as an integral array in the model coordinate system.

2.11.4.1.2. Interaction of Mates

Having applied the positioning mate, the objects shall move automatically in order to satisfy the mate conditions (provided that this condition was not satisfied prior to mate application).

Following the application of mechanical mate, the objects remain fixed (except for the **Cam - Pusher** mate, in which the previously non-contacting objects should automatically be pushed to contact each other).

Mechanical mates work within the "boundaries" defined by positioning mates. The matings' effect can be checked by moving a component in different directions. For more information about mechanical mates see Section 2.11.4.3.6 on p. 718.

While editing a model, mating inconsistencies may occur. For example, you may find that two positioning mates require different positions for the same component. Should this occur, the mating is marked as an “error” in the Model Tree.



If a positioning mate prevents any moves of the component in the direction defined by the mechanical mate, this mechanical mate shall not have any effects. The error symbol shall not appear in the Design Tree.

2.11.4.1.3. Mate Display in the Design Tree



Once a mate is created, it's icon appears in the Design Tree.

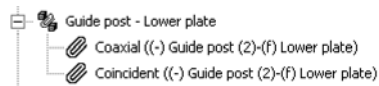


Fig. 2.11.5. Group of mates between Body and Adapter

If the display of the model structure is enabled in the Tree (see Section 2.1.2.2 on p. 106), then two or more mates, created for one and the same couple of components, shall compose a group in the “Mates” Section (Fig.2.11.5). The name of the group is generated from the names of mated components.

2.11.4.2. Positioning matings

As a rule, positioning matings exist in any model, as use of other methods (e.g. displacing the components by mouse, snapping under insertion and other) hardly allows positioning of the assembly components as required, and also, while editing unmated objects, their relative position can be easily broken.

For example, two components were somehow positioned in such a way that two their faces coincided. After changing the extrusion depth of an element belonging to one of the components, the face, with which the face of the latter component has coincided, was displaced. If faces were aligned “manually”, their relative position will be broken. The non-edited component shall remain in it's place. You shall need to reset it into the required position. If coincidence of the faces has been reached by creating components mate of type **Coincidence**, then, after editing one of the mated components and rebuilding the assembly, such a displacement of the latter component shall occur as to keep within the mate condition, so the faces, involved in the mate **Coincidence**, shall be still positioned in one plane.

2.11.4.2.1. General Methods to Create Positioning Mates

To create a mate, call the command corresponding to the required type of mating. Select the mated objects in the model window.

The distance or angle between mated faces is required to be set in commands **At Distance** and **At Angle** respectively.



After selection the objects and setup of all the parameters of mating, confirm its creation.



For the rest of commands, the automated object creation is provided. If the automated mate creation is enabled, the mates are created after the respective objects are specified. The automatic and manual object creation is detailed in Section 1.4.2.10 on p. 93.



If some elements were selected in the Model window before calling the mating command, the mate will be applied to those objects.

Orientation of Components

By default, the mating components are moved in such a way as to satisfy the mate condition, and value of the components displacement, with respect to its original position, shall be minimal. Meanwhile, the position of the components may differ from the required one. For instance, when mating *Coincidence* on the flat faces of parts is done, these parts appear on one side from the plane of the selected faces instead, as the requirement was, of being on different sides from the plane.



To control the position of mating components, disable the mode of automatic confirmation of command **Auto Create** execution on the Special Control Panel).



Then activate one of the switches in the group **Orientation** — **Direct Orientation** or **Reverse Orientation**. Position of mating components can be evaluated by the ghost in the model window.



When the desired orientation of components is achieved, confirm creation of mate.

Additional Methods

- ▼ It is possible to make several similar mates without leaving the current command. For instance, through enabling command **At Distance**, you can place two faces at the specified distance, then, having changed, if needed, the distance specify a vertex and an edge for mating and so on.
- ▼ While mating, you can use command **Remember Status**. For example: you need to position several components in such a way, so that they would touch any surface. After calling the command **Tangency** select this surface, click the button **Remember Status** on the Special Control Panel and select the required components.
- ▼ *Mates can be applied automatically in the process of moving or rotation of the component (see Section 2.11.3.3.1 on p. 700).



2.11.4.2.2. Coincident



To setup coincidence of objects, call the command **Coincident**.

Specify the first and the second objects (faces, edges, vertices, etc. in any combination), which you need to be matched.



Coincidence of coordinate systems of two components means coincidence of axes and planes of the same name of these coordinate systems. Thus, once these components are applied with such mate, they become fixed with respect to each other. Coordinate systems are specified in the Design Tree.

2.11.4.2.3. Coaxial



To setup coaxiality of objects, call the command **Coaxial**.

Specify the first and the second objects (axes, conical faces, etc.), which you want to make coaxial.

2.11.4.2.4. Parallel



To setup parallel of objects, call the command **Parallel**.

Specify the first and the second object (faces, edges, etc.), parallelism of which you want to determine.

2.11.4.2.5. Perpendicular



To set selected objects perpendicular to each other, call the command **Perpendicular**.

Specify the first and the second object (faces, edges, etc.), perpendicularity of which you want to determine.

2.11.4.2.6. Placing objects at specified distance



To position the objects at the specified distance from each other, call the command **At Distance**.

Specify the first and the second objects (faces, edges, vertices, etc.), which are required to position at the specified distance.

Closest Solution

By default, the option **Closest Solution** is enabled on the Property Bar. Therewith, the distance between the selected objects is defined automatically. Mate is created with the use of this distance.

As a result, the position of components after mating does not change or changes minimally.

Thus, for example, if a vertex and a plane were selected to apply an **At Distance** mate, then their positions will not change.

If, however, two planes were selected to apply an **At Distance** mate, then one of them will change its position to become parallel to the other. In this case, the system will choose the closest to original orientation of the displaced component. A mate will be generated using the distance, at which the planes will be situated.

The distance calculated automatically can be changed by the user.

Setup of an Arbitrary Distance

You can set a distance between the mated components prior to specifying objects. To do this, disable the option **Closest Solution** and enter the desired value into the **Distance** field.



To specify, to which side with respect to the first object the distance is plotted, activate the switch **Direct Direction** or **Reverse Direction** in the group **Direction**.

2.11.4.2.7. Arrangement of Objects at an Angle to Each Other



To position the elements at the specified angle, call the command **At Angle**.

To create this type of mating, you can use different objects. Depending on their combination, the mating angle is:

- ▼ for the rectilinear objects— the angle between crossing or intersecting straights containing these objects,
- ▼ for a rectilinear object and a plain object— angle between the rectilinear object and its projection on the plain object,
- ▼ for two plain objects— angle between the planes of the objects,
- ▼ for a rectilinear object and a rotary surface— angle between the object and the axis of rotary surface,
- ▼ for a flat object and rotation surface— angle between the rotation surface axis and its projection on the flat object,
- ▼ for two rotary surfaces— angle between the axes of rotation.



Straight and flat objects are listed in Table 2.1.7 on p. 131. Cylindrical, conical and toroidal rotation surfaces can be used for creation of this type of mating.



There may be two types of mating angles— **3D** and **Flat**.



To select the required type of the mating angle, enable the respective switch in the respective switch in the **Angle type** group of elements— **Spacial (3D) angle** for **Flat (2D) angle**.



When selecting the mating angle type, note that changing the mated components' position will be restricted after mating:

- ▼ if the angle is of the **3D** type— by the value of the angle,
- ▼ if the angle of the **2D** type— by the value of the angle and by its measuring plane position.

3D type of angle

If the **3D** type of angle is selected, specify objects to be mated. The **Angle** field will then become available for entering the mating angle value.

if the **Closest solution** option is enabled in the property bar, the value of the mating angle (angle between specified objects) is defined automatically and entered in the **Angle** field. You can use this or another value to create a mating. In the first case, the position of components after mating does not change. In the second case, the position of one of the components will be changed in accordance with the set value.

If necessary, you can set the mating angle value prior to specifying mated objects. To do so, disable the **Closest solution** option.

2D type of angle

If the **2D** type of angle is selected, specify mated objects and the axis which defines the direction of measuring the mating angle on the plane. The plane of the mating angle measuring (abbreviated— angle plane) is a conditional plane perpendicular to a given axis.

- ▼ **The axis must meet the following requirements:**
 - ▼ The object which defines the axis must be part of one of the mated components.

- ▼ If a component's surface is the mated object, the axis must be parallel to this surface or be part of it; if the mated object is an edge, then the axis must be perpendicular to this edge.
- ▼ **To define axis, the following objects can be specified:**
 - ▼ straight-line object— this object becomes the axis,
 - ▼ flat object— a normal to the flat object becomes the axis,
 - ▼ flat curve (curve, circle, sketch line)— a normal to the curve's plane will be the axis,
 - ▼ rotation surface (cylindrical, conical, toroid)— the rotation surface will be the axis.

For example, the component's mated object is its surface. As an object which defines the axis you can select an edge of the surface.

Once the axis is set, one of the mated components changes its position so that the mating angle sides are parallel to the measurement plane of the angle.

Once the mated objects and the axis are specified, the **Angle** field will then become available for entering the mating angle value. If necessary, you can set the mating angle value prior to specifying mated objects and the axis. To do so, disable the **Closest solution** option on the property bar.

If the **Closest solution** option is enabled, the value of the mating angle (angle between specified objects) is defined automatically and entered in the **Angle** field. Using this value allows to set the mating for mutual position of components to coincide with the initial or there is a minimal difference between them.

You can edit the automatically defined value of the mating angle in the **Angle** field.

2.11.4.2.8. Tangency



To set tangency of objects, call the **Tangency** command.

If you create a mate for those objects having only the *Tangency* mate option, for example, straight edge or cylindrical surface, specify these objects (edge and surface).



While creating objects in automated mode (button **Auto Create** on the Special Control Panel is pressed), a new mate created automatically shall be the closest to the objects' initial positions.

If you mate conical, spherical, toroidal faces, as well as other elements for which several tangency options are available, release the **Auto Create button**.

Specify the first and the second objects (face, edge, surface), which tangency is to be defined.

The following tangency types will be available:

- ▼ by circle— when mating with a toroid, round edge, etc.,
 - ▼ by the generatrix— when mating with a cylindrical or conical surfaces,
 - ▼ in the point— when mating with a spherical surface and tangencing two cylindrical surfaces.
- Once the second object is specified, the object tangency type is determined automatically and either **By Circle** or **By Generatrix** option is enabled. If both of these two options are disabled - the tangency is provided in the point. You can't change status of these options.

Exceptions are the following pairs of objects:

- ▼ *cylinder*— *cylinder* (in the point, by the generatrix);

- ▼ *cone– sphere* (in the point, by the circle).

To set a tangency type for objects of these pairs, enable or disable the respective option.

After selecting objects, the components will be displayed on screen as follows. In case of the by circle **tangency** - the coinciding circles will be in the common plane. In case of the tangency by the **generatrix** - the generatrices will be positioned on the same straight line. In case of the tangency **in the point** - the objects will have a common point.



In a *sphere – toroid* pair, a specific tangency type is possible — in the points of external and internal equatorial circles of the toroid. Such type is considered a circumference tangency.

For plane, cylindrical, cone, non-closed toroidal and spherical faces the tangency line or point may appear on the face extension.



If two mate options are possible, switches of the Orientation **group are available**. To change orientation click either **Direct orientation** or **Reverse Orientation** switch. If more than two mate options are available, button **Next** will be available instead of the orientation switch. After clicking the button, the next option is displayed.



To display other tangency type options for *cylinder– cylinder* and *cone– sphere* pairs of objects, enable the respective option and proceed with viewing.



Once you have selected a mate option, confirm its creation.

2.11.4.3. Mechanical Mates

Mechanical mates ensure relation of components moving in the mechanical transmissions, reducing gears, cam mechanisms and other models. Mechanical mates allow to shift a designed mechanism to various positions by moving one of it's components. While changing this component position, other mated components also move (but with respect to the restrictions applied by positioning mates).

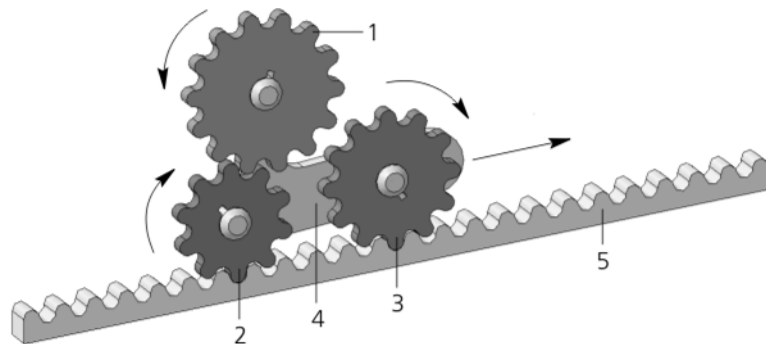


Fig. 2.11.6. Example of parts mated movements

When creating mates, the following issues shall be accounted.

- ▼ Mechanical mates are applied to paired objects similar to positioning mates. The following types of mechanical mates and corresponding movement types are available:

- ▼ **Rotation– Rotation** Both objects rotate.
- ▼ **Rotation– Move** One object rotates, while another — moves along the straight line path.
- ▼ **Cam – Pusher.** One object (cam) rotates while the other (pusher)— moves along the straight path with the contact between the cam and pusher maintained.
- ▼ Several mechanical matings can be created in one model. The same component can be mechanically mated with several components.

In the example illustrated by Fig. 2.11.6, wheels 1 and 2 rotate when wheel 3 is moved along the bar 5. It is ensured by matings **Rotation – move** assigned to wheel 3 - bar 5, wheel 2 - bar 5 and mating **Rotation – rotation** assigned to wheels 1 and 2.

To view operation of a complicated mechanism (if there is no need to show movements of all its parts) it shall be sufficient to setup mates between the first and the last movable elements. In the example, illustrated by Fig. 2.11.6, it might be the **Rotation – move** mate assigned to wheel 1 and part 5. When part 5 is moved, wheel 1 rotates while wheels 2 and 3 do not rotate.



No contact between parts is required to create the **Rotation– rotation**, **Rotation– move** mates and view their operation.

- ▼ When demonstrating mechanical matings, all positioning matings used in setting up a model restrict spatial motions of components. When applying positioning mates, make sure that the components have degrees of freedom to allow for further application and work of mechanical mates (see section 2.11.4.3.1).

2.11.4.3.1. Providing correct work of mechanical mates

To ensure the expected relative relocation of the components bound with a mechanical mate, it is recommended that these components have minimal number of degrees of freedom that are necessary to move within mechanical mates. You can decrease the number of degrees of component freedom using positioning mates with other objects as well as using fixing.

As a rule, if positioning matings are applied in a model so that components have as many degrees of freedom as parts and nodes in a real structure, you do not need to impose additional restrictions to provide work of mechanical matings.

Let us illustrate the above-said by examples.

1. Rotation – Rotation

Let us consider the mate between two gear wheels in which the wheels must rotate about fixed axes (e.g. the wheels 1 and 2 in Fig.2.11.7 on p. 715).

Make sure of the following prior to mechanical mating:

- ▼ the axes are fixed in the model coordinate system,
- ▼ the wheels and their respective axes are coaxial.

Usually, wheels cannot move along rotation axes. You can achieve this by binding flat end faces of wheels to some fixed flat object with the **Coincidence** mate or **At Distance** mate.

So, each wheel will have one degree of freedom to rotate about an axis.

After you apply the **Rotation – Rotation** mate, the rotation of any wheel will result in rotation of another wheel.



Note that wheel rotation axes must be fixed for this particular example. In other cases, axes can move, e.g. satellite wheel axes in planetary drives.

2. Rotation – Move

Example 1. Let us consider the mate between a wheel and bar, in which the wheel must move the bar while rotating (for example, see Fig. 2.11.8 on p. 716).

Make sure of the following prior to mechanical mating:

- ▼ the wheel axis is fixed in the model coordinate system,
- ▼ the wheel and axis are coaxial,
- ▼ the bar is permanently oriented to the wheel axis (you can do this in a variety of ways, e.g. by binding straight edge of the bar to one fixed plane with mate **At Distance** and the other — with mate **Coincidence**).

As in previous example, you can apply the mate preventing displacement of the wheel along the axis.

So, the wheel will have one degree of freedom to rotate about its axis and the bar will have one degree of freedom of translational motion.

After you apply the **Rotation – Move** mate, rotation of the wheel will result in movement of the bar and vice versa - movement of the bar will result in rotation of the wheel.

Example 2. Let us consider the mate between a screw and a nut, in which the screw must move the nut while rotating.

Make sure of the following prior to mechanical mating:

- ▼ the screw cannot move along the rotation axis,
- ▼ the screw and the nut are coaxial.

So, the screw will have one degree of freedom to rotate about the axis and the nut will have two degrees of freedom: to rotate about the axis and move along it.

After you apply the **Rotation – Move** mate, rotation of the screw will result in movement of the nut and vice versa - movement of the nut will result in rotation of the screw.



The screw can be also fixed in this mate. Then, the nut will rotate while moving. But if both the screw and the nut will have two degrees of freedom (to rotate and move), correct work of the mate becomes impossible.

3. Cam – Pusher

Let us consider a mate, in which a cam must move a pusher along the straight path while rotating (for example, as in Fig.2.11.9 on p. 717).

Make sure of the following prior to mechanical mating:

- ▼ the cam rotation axis is fixed in the model coordinate system,

- ▼ the pusher is permanently oriented to its movement path (you can do this in a variety of ways, e.g. by aligning the pusher with the fixed axis if it has circular section).



Working surfaces of the cam and pusher automatically contact each other after you apply the **Cam – Pusher** mate by moving the pusher along its path. Make sure that the shape and relative position of the cam and pusher as well as the mates applied to them do not prevent this. There is no need to apply the **Tangency** mate to the cam and pusher.

We recommend to apply the mates to the cam that prevent its displacement along the rotation axis to keep contact of the cam with the pusher.

So, the cam will have one degree of freedom to rotate about the axis and the pusher will have one degree of freedom to move forward. It does not matter if the pusher has the degree of freedom to rotate about the path.

After you apply the **Cam – Pusher** mate, rotation of the cam will result in movement of the pusher. Inverse movement transfer - from the pusher to the cam - is not provided for in the **Cam Pusher** mate.

2.11.4.3.2. General Methods for Creating Mechanical Mates

To create a mate, call the command corresponding to the required type of mating.

Specify the first of the mated objects in the Design Tree or in the Model window: Specify the object's rotation axis or moving direction path.

Complete the same procedures for the second object.

As for the **Cam – pusher** mate, there are specific issues on object selection. They are described in the Section 2.11.4.3.5 on p. 717.

The specified objects shall be highlighted, and their names shall appear in the respective field of the Property Bar.

Specification of objects, axes and paths is performed after clicking the switch located next to the object field on the Property Bar.



You may cancel currently specified object by clicking the **Specify Other** button on the Special Control Panel.

If required, change relative moving direction of objects. See the following text for more details on how to select directions.

Depending on a transmission type, set the correlation of movements in the **Correlation** field. The value can be input from the keyboard or selected from the drop-down list of correlations previously used in the KOMPAS-3D current operation session.



Once you set up parameters, confirm mate creation.

object selection

Pairs of objects are composed of the components, and one of them can be fixed. In addition, a component can be mated with some other object in the model.

In order to specify rotation axes and movement paths you may select the following objects.

- ▼ Straight lined (segments of polylines, coordinate and auxiliary axes, straight edges).

An axis or path is the line which coincides with the object.

▼ Planar curves (arc, circle-type edges).

An axis or path is the line which passes through the center of the specified object perpendicular to its plane.

▼ Revolution surface faces (cylindrical, cone, toroidal).

The rotation axis is the axis or path.

When specifying some elements along with the object to which these elements belong, the axis or movement path will be selected by default. Its name appears in the corresponding field (**Axis**, **Axis 1**, **Axis 2**, **Path**) of the Property Bar.

For example: an edge of the mated part was specified for its selection for mating. The **Component** field will display the name of this part, and name of the specified edge shall appear in the **Path** field.

Length of displacement along the path is not limited by the specified object length.

Moving Direction

When specifying an axis or path, the system automatically sets the direction of objects movement in the transmission. In the Model Tree, the rotation direction is marked with round arrows, movement/displacement direction - with straight arrow.

Automatically set directions can be changed by group switches **Direction 1** — for the first object and **Direction 2** — for the second.

Only relative direction of mated objects motion is accounted— while viewing transmission each object can be dragged by mouse in both directions - **forward** and **backward**. Herewith, the object moving directions either coincide with the preset one, or are opposite.

For example, if you require wheels of the wheel gearing to rotate towards each other, set opposite directions for these objects (to do this: set switch **Straight Direction** for one object, and **Reverse Direction** for another).



2.11.4.3.3. Rotation– rotation mate

The mate is used to visualize movements in gear, belt, chain, frictional transmissions and other models. To create this mate call the **Rotation - rotation** command.

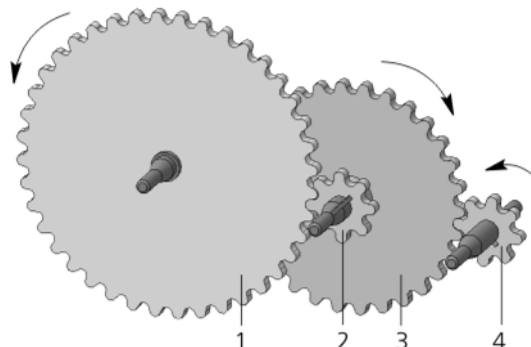


Fig. 2.11.7. Example of Applying Rotation Mates

Specify the first object and the axis to rotate about and then the second object and its rotation axis.

You can specify mated objects by selecting them prior to calling the command.

Set direction and correlation of movements.

Correlation of movements defines ratio of the first object's number of revolutions to the same of the second object. This ratio is determined similar to the gearing ratio. Relation 1:2 means that during one revolution of object 1, object 2 makes two revolutions.

If one of the objects is fixed, the calculations shall be carried out following the same procedure. For example, let one of two mated wheels to be fixed. In this case the moving wheel makes a complicated rotative motion— it rotates about its axis, and at the same time - about the fixed wheel. Correlation shall be presented as the ratio of the number of revolutions of movable axis around the fixed one to the number of revolutions of fixed wheel around it's axis.

When specifying rotation surface, it's radius is entered by default in the **Correlation** field.



You may select parameters for two revolution surfaces by clicking on these surfaces.

The example in Figure 2.11.7 shows how you can apply two rotation mates— between wheels 1 and 2 and between wheels 3 and 4.



It is impossible to apply the mate onto pair of objects on the same rotation axis. Such objects - wheels 2 and 3 - are illustrated in Figure 2.11.7.

2.11.4.3.4. Rotation – movement mate



This mate allows to visualize movements in rack-and-gear transmissions, screw-and-nut assemblies and other models. To create the mate, call the **Rotation – move** command.

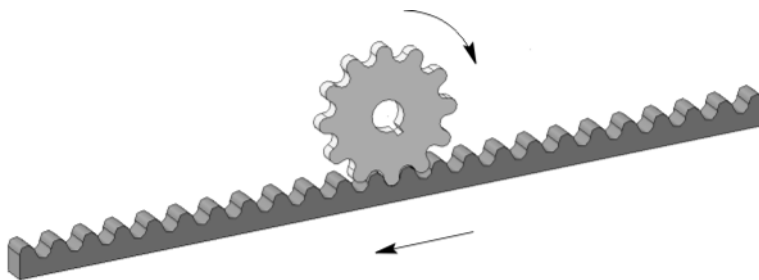


Fig. 2.11.8. Example of use of the Rotation– movement mate

Indicate the first object and the axis to rotate about and then the second object and the path specifying direction of its movement.

You can specify mated objects by selecting them prior to calling the command.

Set direction and correlation of movements.

Correlation of movements defines the ratio of the single radius circumference (2π mm) to linear displacement of the second object.

Relation 1:2 means that during one revolution of object 1, object 2 moves by $2 \cdot 2\pi$ (mm), i.e. about 13 mm.

If one of the objects is fixed, the calculations shall be carried out following the same procedure. For example: let the bar is fixed, the wheel of radius R (mm) moves along it without sliding. During one revolution of the wheel the bar moves by $2\pi R$ with respect to the wheel center. Displacement ratio is 1: R .



You can calculate the required correlation using by the following formula: $1:(L/2\pi)$.

Where L — is a distance in mm the second object will pass per one revolution of the first object.

2.11.4.3.5. Cam – pusher Mate



The mate is used to visualize movements in cam mechanism. To create this mate, call the **Cam – pusher** command. The cum makes a rotary motion, while the pusher goes backward and forward. Here the pusher operating surface remains contacting the cum operating surface.

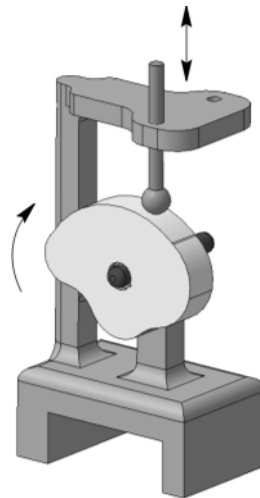


Fig. 2.11.9. Example of the Cam-Pusher Mate

Mated Objects

The mate objects shall be selected so as one of them would function as a cum, rotating about the axis, while another shall be a pusher moving along a straight path. The mechanism structure shall ensure that cam contacts pusher within their respective working surfaces.

The **cam** model shall be a single-body part.

The cam's working surface may comprise one or several faces— flat, cylindrical, cone, toroidal. Moreover, the cam working surface may look like a ruled surface, whose sketch consists of lines, arcs, ellipse arcs, splines.

In order to specify a **cum rotation axis**, you may select the following objects.

- ▼ Straight lined (segments of polylines, coordinate and auxiliary axes, straight edges).
An axis is the line which coincide with the object.
- ▼ Planar curves (arc, circle-type edges).
An axis is the line which passes the specified object center along the normal with respect to its plane.
- ▼ Revolution surface faces (cylindrical, cone, toroidal).
The rotation axis is the axis.
The **pusher** working surface is a flat or spherical face. The mate process engages the surface which has been generated by extending the specified face— plane or full sphere.
The pusher part top level can function also as the working surface,.



Should working surface of the pusher be flat, working surface of the cam can be a cylinder, cone or ruled surface only. Therewith, the cum and pusher shall be mutually oriented so that they could contact each other along the straight line.

Straight objects (segments of polylines, coordinate and auxiliary axes, straight edges) can be **specified as the pusher paths**. The pusher will move along the selected path.

Mate Creation



While the Cam **Faces** button is pressed, specify one by one those faces of the cam working surface, that shall be used for mating. Number of faces appears in the **Cum Faces** field.



Click the **Axis** button and specify the axis the cam will rotate about. It's name shall appear in the axis field.

If faces of the cum working surfaces are smooth-mated, selection of one of the object faces, while the **By Tangent** option is enabled, results in automatic selection of adjacent faces. Only those faces will be selected that are joint along fragments of straight lines, which are parallel to the cum axis.



Click the **Pusher Face** button to specify a working face or pusher top.



Click the Path **button and** specify the pusher movement path. It's name appears in the path field.

Once an axis or path is specified, the system shall automatically determine direction for rotation or displacement along the path. In the Model Tree, the rotation direction is marked with round arrows, movement/displacement direction - with straight arrow. The cum and pusher working surfaces start contacting.

2.11.4.3.6. Viewing Mates operation

To check or visualize mate operation, move one of mated components. Such movement shall comply with those parameters preset during mate creation. In other words, you need to select an axis to rotate and the direction preset during mate creation to move. This movement shall *initiate* displacement of the mated object— *response* displacement. In it's turn, the object's response displacement can initiate displacements of other inter-mated objects, etc.



To initiate moving a component, use the commands listed below:



▼ **Move Component,**



▼ **Rotate Component About Axis,**

▼ **Rotate Component About a Point,**



▼ **Rotate Component About the Central Point.**

For more information on model component moving commands, see Sections 2.11.3.1 – 2.11.3.2.

The initiating movement can be a part of a component's complex movement. For example: to view mate **Rotation–move** you may use the command **Rotate Component About Axis**. While dragging the bar by mouse (see Figure 2.11.8 on p. 716), the linear component of bar rotation shall serve as initial displacement. Only that linear component is accounted, which is co-directed with the mate-preset path.

If movement of an object is not initiating and has no initiating components, the mated objects shall not move in response.



You can view only those matings of the current model that are applied to this model's components and bodies.

When viewing mates, keep in mind the following conditions.

- ▼ Positioning mates can prevent change object positions.
- ▼ In order the rotary motion to be initiating, make sure that the axis, selected to complete this task, coincides with the axis preset in mechanical mate. If this axis is set automatically in the mate during specification of an object element, construct such axis as, for example, an auxiliary object.
- ▼ Direction of the initiating movement can either coincide to that one preset in the command or be reverse. All parts of mechanical transmissions shall move in accordance with relative directions preset during mate creation.
- ▼ A component's rotation or movement along the path is not limited. To avoid “penetration” of one model component to another, you may set your preview mode to the collision detection. For more information about collision detection mode, see Section 2.11.3.3.2 on p. 700.
- ▼ If any object, which sets displacement direction (e.g., edge), belongs to a movable component, the path shall change its position in space to comply with the mate conditions.
- ▼ You can create mate of movable and fixed (non-movable) objects. In this case, a movable object - if its movements are not affected by positioning mates - will do complex rotary or rotary-translation motion with respect to non-movable one.
- ▼ To preview the **Rotation – move** transmission it is important to set a relation which shows real displacement of objects. The relation calculation method is presented in Section 2.11.4.3.4 on p. 716.
- ▼ Preview of the **Cam – pusher** transmission is only possible when moving the cam. Mated move takes place only within the faces specified as its working surfaces. The cam faces, not specified in the mate, may lose contact with the pusher.

- ▼ While previewing the **Cam – pusher** transmission, the contact may be invisible since the pusher working surface is the entire spherical surface or edge plane.

2.11.5. Boolean operations on Parts

The parts (including local parts) constituting one assembly (*.a3d) can be “subtracted” from each other and “attached”.

Boolean operations with parts are performed when editing them in an assembly.

The result of a Boolean operation of subtraction is modifying the body of the part by removing cross-sections with the bodies or components of other parts from them. Components of a part are not modified by Boolean subtraction even if they cross-section with the parts being subtracted. The subtraction result is stored in the part from which other parts have been subtracted.

The result of Boolean operation of addition is a body that encompasses the bodies of other parts or their components. The result is written to a new file of the part or to a new local part.



A body can take part in a Boolean operation only in case the part it contains includes no other bodies. Components can be used in a Boolean operation regardless of their number in the part.

The result of a Boolean operation preserves connection with parts used in the operation. For example, changing the form of a part being subtracted will result in a changed form of the hollow area that results from the subtraction. Removal from the assembly of a part involved in a Boolean operation will result in the deletion of the operation, too.

2.11.5.1. Subtraction

Subtraction allows a cavity to be created in the part that has the form of another part. To subtract one or more parts from a part, enter context editing mode for the part you want to have a cavity as a result. To do so, select the part on the Design Tree and call the command **Edit in place** from the context menu.



Call the command **Operations — Subtract components**. The respective command calling button is located on the panel **Edit Part**.

Select the parts you need to subtract from the edited one.

Selected parts are highlighted in the Model window. Their respective icons will be highlighted in the Design Tree. The names of these parts are displayed in Table **List of Components** on the Property Bar.

If it is required that the sizes of the cavity to be created differ from those of the part to be subtracted, enter in the field **Coefficient** the cavity linear extension coefficient in percentage terms. To enlarge the cavity sizes, the coefficient value should be positive, to reduce — negative.

The cavity will be increased in relation to the subtracted part by $(1 + k/100)$ times, where k is specified coefficient. By default, the void scaling center is the outline box center of the part being subtracted. If necessary, you can explicitly define the scaling center. To do this, select the

desired point in the Model window with the mouse. To reject this point as the center of scaling, select it again.



Confirm execution of the operation. In the current part the cavity, having the specified shape and sizes, will be generated.



The subtracted part can have such a position and shape that after subtraction a body consisting of several parts can be produced (see Section 2.3.11 on p. 224).



On the “branch” of the Design Tree corresponding to the current part there will appear the icon of the components subtraction operation.



When finished with creation or editing the part, exit from the mode of context editing.

If necessary, you can hide or exclude from calculation the parts, used for the cavity creation.

2.11.5.2. Union

While working with the assembly, you can “merge” several existing parts and get a single part. For example, this can be required for union of the designed parts into a single cast framework subject to the new specifications arisen in the process of designing.



To unite several parts, create a new part in the assembly. To do so, use the **Operations — Create** menu to call the command **Part** or **Local part**.



The system enters the part editing mode.



Call the command **Operations — Unite Components**. The respective command calling button is located on the panel **Edit Part**.

Select the parts, which are to be united. To perform the operation, the selected parts need to intersect each other or have coincident faces.

Selected parts are highlighted in the Model window. Their respective icons will be highlighted in the Design Tree. The names of these parts are displayed in Table **List of Components** on the Property Bar.



Confirm execution of the operation. In the model window, there will appear the body of the new part which is the union of the selected assembly parts, and on the Design Tree — the corresponding icon.

Methods to work with the derived solid body do not differ from work with any other part. You can attach and cut the shape-generating elements, create fillets, drafts and so on.



When finished with creation of the part, exit the mode of context editing.

If necessary, you can hide or exclude from calculations the parts that were joined into the new part.

2.11.6. Editing components

Components are independent models inserted in the current model.

To change the geometry of a component you need to either open it in a separate window (see Section 2.11.6.1 on p. 722), or enter the context editing mode for the component (see

Section 2.11.6.2 on p. 723). This done, you can now use any editing techniques described in Section 2.14.3 on p. 786.

Please note that editing in a separate window is available for components that are a part, sub-assembly, or base part inserted with history. Local part and base part without history can be edited only in the context of the current model.



Upon completing editing the component, close its window or exit context editing mode. Edited components are normally marked with a red tick mark on the Design Tree. This is an indication that the model needs to be rebuilt. To rebuild the model, call the command **View — Rebuild**. For more details about rebuilding models, see Section 2.14.3.7 on p. 795.

If required, you can edit the properties of the component (see Section 2.11.6.4 on p. 725). When editing, the following operations are available:

- ▼ changing component properties, including color and optical properties,
- ▼ fixation control and component state (including the component into the calculations),
- ▼ setting parameters of the component's MP and conducting calculations,
- ▼ selecting another file as a component source,
- ▼ re-calculating the component's dimensions taking into account preset tolerances.

In addition, you can change the position of the component in the model (see Section 2.11.6.3 on p. 724). The following actions are available within this process:

- ▼ setting the component's position using different methods,
- ▼ fixation control and component state (including the component into the calculations),
- ▼ changing the color and optical properties of the component,
- ▼ specification object creation control,
- ▼ selecting values of the component's external variables.

2.11.6.1. Editing the geometry of the component in a window



To start editing a component in a separate window showing this component only (without other components), select the desired component in the Design Tree and use **Edit in window command**. It is recommended that before editing you save the current model.



You can proceed with editing a component in the window by opening its file in the usual way: use **File — Open** command and select the desired file. When using the **Edit in Window** command, you do not need to search for the desired file in the list — the system finds it automatically.

After calling the **Edit in window** command, the source file of this component will open. Make the desired changes to the model and save it. Then close the component window or go into the model window.

When editing a component “in window”, all objects that belong to this component are displayed and are available for editing (you can modify object parameters, their reference objects, etc.). However, editing certain elements is restricted. This includes component objects that were created in the model context using elements from other components of this model (i.e.

using elements that do not actually belong to the component being edited), such as component's auxiliary plane running through edges and/or vertices of other parts, etc.

In the component file that contains objects defined in the assembly context there are links to the file of the model (those links are called external since the assembly file is **external** with respect to the component file). When editing such a component in a separate window, the only available means is to redefine the external links to make them internal. In other words, to construct an object referencing objects from an external file, you can specify other objects available in the current file. An element edited in this way loses connection with the external file.

However, if it is required that the element defined in the model context does not lose the relation with it, then you need to edit it in the context of that model ("in place").

2.11.6.1.1. Saving component under other name

If both the model file and the source file of a component are opened simultaneously, then the system will request the user to change the link to the component when trying to save the component file under a different name or in a different folder.

- ▼ If you want the model to refer to the component file under a new name, click the **Change links** button. This may be needed, for example, for including an edited component into a model and saving its original instance.
- ▼ If you want the model to refer to a file with the old name, click the **Save links** button. This may be needed, for example, to save an edited component without transmitting its modifications to the model that contains it. Note that in this case the model will have to be closed without saving after component editing is finished.



The links are changed only in the currently open models. The links are not changed in the models that are linked to this file but are closed at the time of its saving under a different name.

2.11.6.2. Editing component geometry 'in place'

Editing "in place" (in the model context) is editing a component in the window of the model to which it belongs. At the same time, all the rest of the components are shown in the window ("the environment" or "the situation"), and the commands to shift and rotate the model, change its scale, orientation and display type are available.

It is especially convenient to edit "in place" the component objects that were constructed using "environment" (for example, an element extruded up to a face of a neighboring part, or a construction axis passing through the coordinate origin of another component). In other words, the objects that were created in the model context should preferably be edited in the context of the same model.



When editing 'in place', the updates are made directly to the component file (part or subassembly) — just as it would be if the file was opened in a separate window. The prompt to save the component file appears when editing is complete.

When editing a local part or base part without history the save file prompt does not pop up, as these are stored not in an external file, but in the file of the model that contains them.

2.11.6.2.1. Context editing mode operation



To start editing an assembly component in the current window, select the desired component in the Model Tree and use **Edit in place** command. Button for this command is on the **Current Status** bar.

The system will enter component editing mode.

The icon of the component being edited in the Model Tree becomes orange instead of blue, and the icon of the containing model becomes vice versa blue instead of orange. This indicates that the component has become the current document.

The active “in-place editing” mode is indicated by the pushed **Edit in Place** button, and the context editing mode icon in the model window.

In this mode, all construction and editing commands affect only the selected component. Other model components are not accessible for editing (serve as “background”), but they can be used when executing commands (to select faces, edges, vertices).



You can change colors for displaying components which make up the “environment” or disable change of color for these components (see Section 9.1.11.8 on p. 1732).

Techniques for creating/editing models 'in place', in the context of containing model, are virtually no difference from the ones for creating/editing models in a separate window. When working with a model you can, for example, perform shape-generating operations, create auxiliary elements, add components from file, create components 'in place', etc. Additionally, you can use it to create 'environment' elements. For instance, you can extrude a shape-generating element up to the facet of the other component, draw an axis through its vertices, create an outline on the face of an adjoining part (when working with an assembly).

To complete editing the component 'in place', use **Edit in place** command once again or release the **Edit in place** button, or click the context editing mode icon in the model window.

The system will return to main model operation mode.

2.11.6.3. Changing the position of a component



To change the position of the component in the model, select it on the Design Tree and call the command **Position Component** from the context menu or **Edit** from the **Editor** menu. The **Location** and **Properties** tabs will appear on the Properties panel. The following actions are available on these tabs.



- ▼ The **Location** tab
 - ▼ Changing the component's position in the model (see Section 2.11.2.5.2 on p. 692). The set of component positioning control elements depends on the chosen positioning method — **By coordinates** or **By matings**. The elements are available if component fixation is disabled.



If a component whose position needs to be changed is positioned in the model using the **By matings** method, note the following peculiarity.

If you switch to the other positioning method (**By coordinates**) previously created matings are not deleted from the model, but are excluded from the calculations.

If necessary, you can include the matings in the calculations. In doing so, the component's position will change accordingly to the set matings.



When editing a base part inserted without history, note that such a base part does not keep links with the source file. For this reason, if you set its position using the **By matings** method the model window does not show the source model screen.



▼ Turning on/off components fixation using switches **Fix component** and **Do not fix component** in the **Fixing** group.



▼ **Properties** tab

▼ Changing the color and optical properties of a component. Control elements used for this are described in Section 2.14.2 on p. 784.



▼ Turning on/off creating a specification object using respective option (for all components, save base parts).



▼ Including component into the calculations and excluding one. Switches in the **Status** group are used for this.



▼ Selecting new values of external variables of the model that is the source of a component, if it contains a table of variables. For this, you need to click the **Table of Variables** button and select the required row in the table that appears on the screen.



To finish changing the position of a component, click the **Create object** button on the Advanced Control Panel.

2.11.6.4. Changing component properties



To change component properties, select it on the Design Tree and call the command **Component properties** from the context menu. The **Properties**, **MP parameters** and **Source file** tabs will appear on the Properties panel. The following actions are available on these tabs.

▼ **Properties** tab

▼ Changing component properties using the **List of properties** panel (see Section 5.1.4.1.1 on p. 1385).



▼ Changing the color and optical properties of a component. Control elements used for this are described in Section 2.14.2 on p. 784.



▼ Turning on/off components fixation using switches **Fix component** and **Do not fix component** in the **Fixing** group.



▼ Including component into the calculations and excluding one. Switches in the **Status** group are used for this.



▼ **MP Parameters** Tab

- ▼ Selecting MP determination method and setting component material properties using the **Material** panel (see Section 2.14.6.3 on p. 805).
 - ▼ MP recalculation. Is carried out by clicking the **Recalculate MP** button, which results in recalculating mess-center characteristics of the component with current parameters. The Message window with the calculation results pops up when calculation is finished.
 - ▼ **Source File** Page
 - ▼ Selecting another model for insertion, as well as the model's version or additional number (see Section 2.11.6.4.1).
 - ▼ Re-calculating the component's dimensions taking into account tolerances. To do so, click the **Recalculate dimensions** button and select the required recalculation from a recalculation drop-down list (more about recalculations, see Section 2.10.3.1 on p. 672).
- To finish changing the component's properties, click the **Create object** button on the Advanced Control Panel. The model image will be redrawn and/or the model will be redrawn according to the new properties of the object.
- Note that the model has to be rebuilt if:
- ▼ the component's source file is modified, its other version or additional number is chosen,
 - ▼ component's dimensions are changed taking account of tolerance fields.



2.11.6.4.1. Changing component source

If you need to select another model as the source of the component, take the following steps on the **Source file** tab.



Click **Source file**.

Select the file, containing the required model, in the dialog appeared on screen.

If the selected model contains versions and additional numbers, you can select the required version and additional number using appropriate lists. Lists are displayed in the dialog provided the model contains versions and additional numbers.

To complete selection of the model, click **Open**. The full name of the model will appear in the **Source file** field.

If will need to reselect the version or additional number of the model later on, use the **Version** and **Additional number** lists on the **Source file** tab. Lists are displayed in the dialog provided the model contains versions and additional numbers.



For a local part or base part inserted without history you cannot select a different model, and you can select model version and additional number only in case if the insertion source was a model that contains versions and additional numbers.

2.11.6.5. Particular features of working with local parts and base parts without history

There are the following peculiarities of working with a local part and base part inserted without history as compared with other components.

- ▼ The user can control displaying objects in the Design Tree which make up the local part (base part without history). To do so, the following commands **Show structure** and **Hide structure** in its context menu. By default, a newly created local part is displayed, and base part without history is not. Controlling the display of the composition of a local part (base part without history) is similar to that of the composition of a macroelement (see Section 2.15.8.2 on p. 840).
- ▼ Local part and base part without history are not linked with **.m3d* files, but are stored in the current model. For this reason, they can be edited only in the context of the main model. The context menus of these components contain only the **Edit in place** command, while the **Edit in window** command is not available. Editing a local part and base part 'in place' is the same as editing other components (see Section 2.11.6.2 on p. 723). The only difference is as follows: when exiting the edit mode there will be no prompt to save the file.
- ▼ Local part and base part without history are always fully loaded, with **Full** loading type setup only available for them.
The remaining work techniques — shift and rotation, mating, setting properties, access control, etc. — are available as much as when working with other components.

2.12. Assembly Peculiarities

2.12.1. Background KOMPAS

A KOMPAS-3D assembly is a three-dimensional model combining models of parts, subassemblies and standardized parts along with the information about the relative position of components and dependencies between parameters of their elements.

The user defines the assembly composition by adding new components or deleting existing ones. For more details on components, see Section 2.11 on p. 683.

Two approaches to designing an assembly are used, tailored to two methods of adding a component to an assembly:

- ▼ **Bottom up** — ready component models developed independent of each other are inserted in the assembly;
- ▼ **Top to bottom** — the components are created in the context of the assembly, and the subsequent ones can be based on the preceding ones.

In real life, a mixed design method is used, i.e. a combination of the said two methods. In addition, **composition geometry** can be used— baseline data presented in the graphics form and used as a basis for creating the geometry of the components, or an assembly 'layout' of sorts, used to place the components. If you need to model movements of assembly components, i.e. show the workings of the mechanism, composition geometry can be used a mobile wireframe that set in motion the components that are linked with it.

For more details about design approaches, see Appendix VII. Design methodologies used within the approaches are discussed there as well. About working with composition geometry, see Section 2.8.6 on p. 618.

If the assembly is large enough, i.e. includes over 1000 components, it is usually developed by several developers. In addition, large assembly components often have rather complex geometry.

In connection with this, the following must be provided during work with large assemblies:

- ▼ optimized use of computer resources,
- ▼ control of access to various assembly parts when several developers work with it,
- ▼ protection of working results of one assembly developer from modifications of another developer.

The above-listed requirements can be satisfied by:

- ▼ special settings KOMPAS-3D that reduce consumed computer resources (see Section 2.12.1.1 on p. 730),
- ▼ banning changes to some of the assembly's components (see Section 2.12.3 on p. 744),
- ▼ creating assembly load modes (see Section 2.12.2.2.2 on p. 736) and protecting them with passwords (see Section 2.12.2.3 on p. 741).

Recommendations on how several developers can work with an assembly are given in Appendix VII.



The model file internal structure has been improved starting from Version 10 and higher of KOMPAS-3D. In this context, if the model is created in the KOMPAS-3D Version 9 or lower versions, we strongly recommend to save the assembly and its components in the current version.

2.12.1.1. Settings of the 3D editor to work with a large assembly



1. When working with large assemblies or use the simplified display mode for faster offset and rotation of a model. Set up the simplified display mode as follows.
 - ▼ In the dialog which is used to simplify components, reduce detalization level and enable auxiliary mode for the standardized components. If your model comprises a large number of standardized components, we recommend to disable display of these components by the **Hide option**.
 - ▼ Enable all options in the dialog which is provided to set up other parameters of the simplified mode.

For more information about display of models in simplified mode, see Section 2.1.3.6 on p. 129.



The model simplified display mode is enabled by default; there are also enabled all options in the dialog which is used to set up simplified display mode.

2. Disable all options in the Orientation Settings dialog (see Section 9.1.11.6 on p. 1731). This allows to exclude additional rotations and scaling of the model.
3. Make sure that the **Display Read-only status for components** option in the access rights setup dialog is disabled (see Section 9.1.4.2 on p. 1694). If the option is enabled, more time for the model's file opening is required due to performance of a check for components' read-only status.
4. To reduce time for model redrawing after shift or rotation, make the following settings.
 - ▼ Using commands of the **View — Hide** menu disable display of those assembly auxiliary objects that are not required now. Display in the assembly of major part of the object auxiliary components is disabled by default. If required, you may hide the rest objects using commands in the **View — Hide in Components** menu.



- ▼ Disable display of half-tone wireframe images.
5. To restore quickly the frequently used model positions, save them as customized orientations (see Section 2.1.3.2.1 on p. 123).

2.12.1.2. Settings for large assembly's associative drawings

To speed up construction of associative assembly views in a drawing, it is recommended to proceed in the following way: prior to creating associative views, make the following setting for the current drawing in the new view settings dialog (call the respective dialog using the command **Service — Parameters... — Current Drawing — Document Parameters — View**):

- ▼ on the **Parameters** tab: disable options **Hidden** and **Library**,
- ▼ On the tabs **Lines** and **Dimensions and Process Symbols**: disable all options.

Such setup shall allow you to get on the drawing only model visible contour lines.

Images of cosmetic threads, as well as images of hidden and library components can be enabled at the final stage. Cosmetic threads are enabled on tab **Dimensions and Process Symbols** of the Property bar (insertion is possible during editing); component images are enabled using the **Show** command from contextual menu for these components in the Drawing Tree.

Displaying of hidden lines, tangent edges and bend lines can be activated in tab **Lines** on the Property Bar (activation of this displaying mode is possible during view editing). However, when working with assemblies comprising over 1000 parts, it is not recommended to enable transfer to the view of hidden lines, otherwise it would essentially increase the time required to construct and reconstruct views.



The model file internal structure has been improved starting from Version 10 and higher of KOMPAS-3D. In this context, if the assembly model is created in the KOMPAS-3D Version 9 or lower versions, we strongly recommend to save the assembly and its components in the current version. This should speed up construction of views and transfer of mass value from the model file to the drawing title block.

2.12.2. Partial assembly loading

Partial assembly loading means temporary presentation of chosen components in a simplified, unloaded view or as outline boxes.

The user assigns loading types to components (see Section 2.12.2.1.6 on p. 733) and thus creates the assembly loading type. Assembly loading types are different combinations of components' loading types saved in the assembly (see Section 2.12.2.2 on p. 735).

Bodies which are built in an assembly and local parts contained in it have no loading types. They are always entirely present in the assembly.



Any changes made in an assembly loading type are regarded as editing the assembly; therefore, when the assembly is being closed, the request for saving the edited file will appear. Select **Yes** to save assembly files and all assembly components, or **No**, if you do not need to save those documents.

2.12.2.1. Component Loading Types

Component load modes are component load methods differing by data amount stored in computer memory: **Full**, **Simplified**, **Empty**, **Dimensions**.

2.12.2.1.1. Full

If the **Full** loading type is selected, the component is loaded in a normal way, i.e. it will be visible in the model Tree or window. The **Full** loading type is not labeled by a special symbol in the Tree.

By default, all created components have the **Full** loading type.

2.12.2.1.2. Simplified

In case of the **Simplified** loading type, a component is loaded so that it is displayed in the Design Tree and Model window. However, in the Model window, the components are displayed without wire frame lines and with less accuracy (i.e. less “smooth”).



A simplified component icon is marked with the “cloud” symbol,

See Section 2.12.2.1.5 on p. 732 for more information about partial loading components.

2.12.2.1.3. Empty

In case of the **Empty** loading type, the component is not loaded. It can only be seen in the Design Tree.



Half-tone gray square is applied on the unloaded component icon,

See Section 2.12.2.1.5 on p. 732 for more information about partial loading components.

2.12.2.1.4. Dimensions

In case of the **Dimensions** loading type, the component is loaded so that it could be seen in the Design Tree, while only its outline box is displayed in the model window. Refer to Section 9.1.11.13 on p. 1737 for more information about the models' dimensions display settings.



The icon of the component with the **Dimensions** loading type is denoted with the “3D box” sign.

See Section 2.12.2.1.5 on p. 732 for more information about partial loading components.

2.12.2.1.5. Peculiarities of partial loading components

Components with partial loading (**Simplified**, **Empty**, **Dimensions** types) have the following differences from the fully loaded components (**Full** type):

- ▼ preview of the component composition is not available (no “+” to the left from such component name),
- ▼ it is impossible to view component's variables (the list of component's variables is not present in the Variables Window),
- ▼ the BOM internal object, corresponding to the component, is not available for editing in assembly.

It is also the unloaded components' (**Empty** type) peculiarity that, if just one such component is a part of a mating, the mating is **blocked**. Such blocked mating cannot be edited or deleted. Such blocked mating is marked by a lock icon in the Design Tree. To remove blocking of the mating, all participating components should be fully loaded.

The partial loading component loads automatically in full when the following actions are performed:

- ▼ any operation is applied to the component (creation of a round hole on the component's surface, rounding of the component's edges and so on), as well as editing of this operation;
- ▼ specifying a component for construction of another object (specifying the component's surface for creation of a sketch or thickening and so on), as well as editing the object;
- ▼ mating for a component, as well as editing the mating;

- ▼ specifying the component as an initial object for an array of components, as well editing the array;
- ▼ movement of a component within an assembly's structure;
- ▼ component shift or rotation.
- ▼ calling the component editing command in the window or in place;
- ▼ changing the component's source file.

Hereafter, the automatically loaded component can be unloaded, simplified or dimensions can be applied to it. However, bear in mind that full loading of a component should be automatically restored when the assembly is re-built. This occurs in the following cases:

- ▼ component is a source object or an instance of an array of components,
- ▼ an operation is applied to the component which results in creation or removal of surfaces and edges (**Chamfer**, **Hole**, **Surface splitting**, **Remove surfaces** and so on).
- ▼ subassembly component is loaded in full if model diversification commands are executed, except for the **Simplified** component loading type when components remain simplified after its diversification or canceling.

2.12.2.1.6. Selection of the component loading type

You can select a loading type for a component at any assembly level.

In order to change loading type of a selected component (part or subassembly), call the **Loading types — Full/Empty/Simplified/Dimensions** command (Fig. 2.12.1) from the context menu or from the **Assembly editing** bar.

You can select a loading type created in the assembly's source file for a subassembly component. To do so, highlight the subassembly component and run the **Loading types — Select...** command from the context menu or click the **Loading type applied** button on the **Current state** bar. For details about assembly loading types see Section 2.12.2.2.3 on p. 739.



Loading type can be changed for several components of the same type, all objects of the *Components* section, all subassembly components, as well as components group or branch. To do so, select components, section, group and so on, and call the command as it is described above.

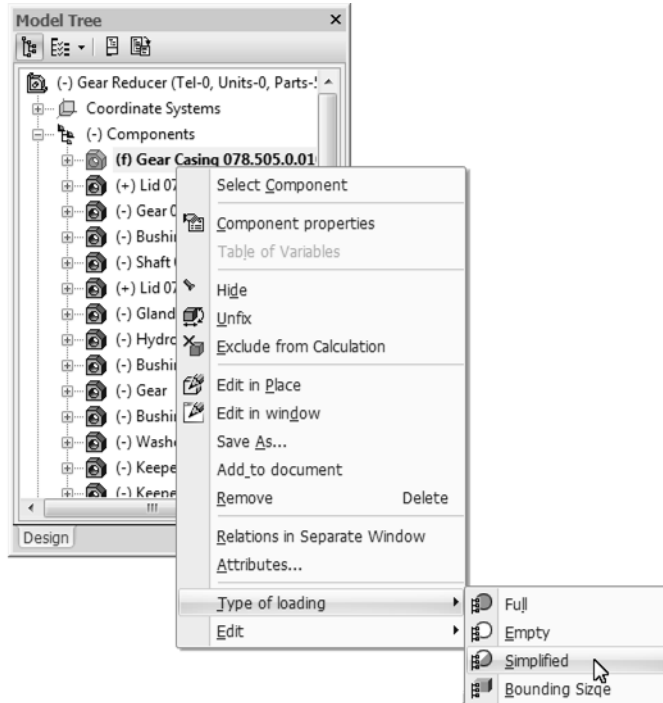


Fig. 2.12.1. Selecting a Component Loading Type



Alternation of a loading type for one array instance should be applied to other instances and initial component.

Changing a loading type of one component insertion (including a standard item) engages all other insertions of the same component in the current assembly.

If loading type is changed, note the following.

- ▼ Loading types with descending detalization — **Full, Simplified, Dimensions, Empty**.
You can change the loading type for components to the **Full** type or to a less detailed type.
For example, components with the **Empty** or **Dimensions** loading type cannot be loaded with the **Simplified** type. In order to do such loading, you should at first load the components in full and then partially.
- ▼ You cannot apply a loading type which is less detailed than that applied in the component file, if the file is opened in a separate window. For example, a part component cannot be unloaded, if its source file is open.
- ▼ You can apply any loading type created in its source file to a subassembly component. If several subassembly components are selected for execution of a command, note the following:
 - ▼ loading types of the first specified subassembly element are available on the Property Bar;

- ▼ loading type is applied to all selected subassembly components which have a loading type with the same name in their files; each subassembly is loaded individually in accordance with the loading created in the process of this loading type creation.



Subassembly components are links to their respective files.

Therefore, if model files are opened, any changes of loading types (for example, creation, editing, or applying) in the subassembly file are transferred to the assembly which contains it. And, vice versa, if a subassembly has a different loading type in assembly, it will also be automatically applied in the subassembly file.

2.12.2.2. Assembly loading types

Assembly loading types are different combinations of assembly components' loading types (see Section 2.12.2.1 on p. 731).

There are **system** and **user** assembly loading types.

Every assembly has system load modes accessible: **Full**, **Empty**, **Simplified**, **Dimensions**. When you select one of them, all assembly components will be loaded in the same mode (Fig. 2.12.2, a).

User loading types are created for an assembly by the user and saved in the assembly file. User types are used to load various components in different ways and/or prohibit their editing (Fig. 2.12.2, b). See Section 2.12.2.3 on p. 739 for information about application of loading types.

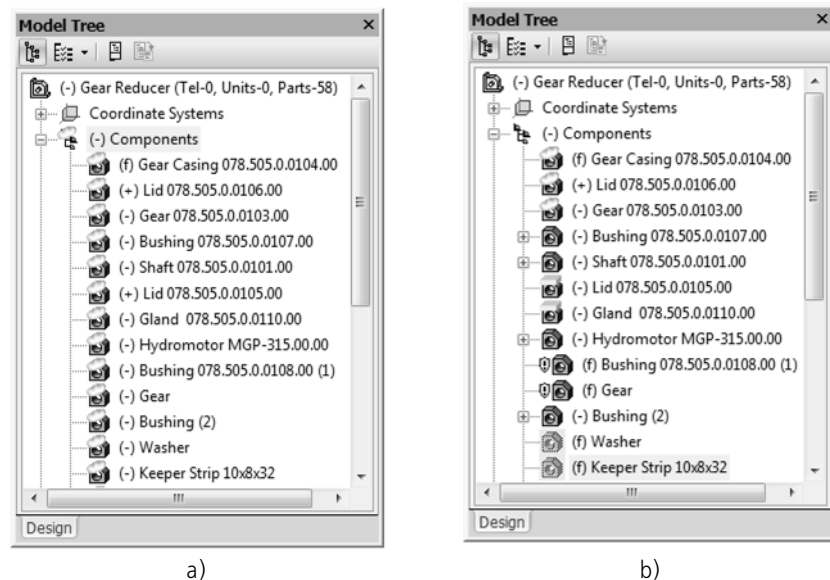


Fig. 2.12.2. Assembly loading type examples: a) simplified system type, b) user type

One of load modes in an assembly is **applied**. By default, this is the *Full* system load mode. The applied assembly loading type is displayed on the **Current state** bar. An asterisk in front of the loading type (Fig. 2.12.3) means that the loading type of one or several components is

different from the one applied (i.e. was changed by the user, see Section 2.12.2.1.6, or automatic change from partial to full loading took place, see Section 2.12.2.1.5 on p. 732).



Fig. 2.12.3. Displaying the applied loading type

Using of simplified loading types allow to free resources of your computer and speed up work with large assembly models. Thus, you can select the **Full** load mode only for the subassembly with which you are working at the moment and the **Empty** load mode for the rest components. If you need to see location of these components in an assembly, they can be loaded in **Simplified** or **Dimensions** mode.



Partial loading of components is preferred to excluding components from calculations, these loading types do not affect hierarchical and associated relations between model objects. Besides that, components with the **Simplified**, **Empty**, and **Dimensions** loading type, in contrast to those excluded from calculations, are transferred to the associative drawing of the model in the same with as those with the **Full** loading type.

Any assembly load mode can be password protected (see Section 2.12.2.3 on p. 741).

2.12.2.2.1. System loading types

System assembly loading type — **Full**, **Empty**, **Simplified** or **Dimensions**. Applying one of these types means that all assembly components have the same loading type. For example, the **Simplified** loading type means that all of its components are downloading with simplification. For details about component loading types see Section 2.12.2.1 on p. 731.

By default, the **Full** loading type is applied to assemblies.

System types cannot be deleted from the model.

2.12.2.2.2. User loading types. Creation and deletion

The custom assembly load mode is a certain combination of load modes of assembly components. Moreover, the custom load mode allows to save information on “read-only” attributes of components.

You may preset several customized loading types for the same assembly. All customized loading types are stored in the assembly file.

Password protection of assembly load modes allows to control access to nodes of a large assembly in case of team work, provided that different custom modes deny access to some components and allow access to others (see Appendix VII).

To switch to managing loading types, call any of the following commands:

- ▼ the **Loading types** command from the assembly's context menu;
- ▼ the **Loading type applied to assembly** command by clicking the same-named button on the **Current state** bar.



Once the command has been called, control elements appear on the Property Bar making it possible to create, edit, delete loading types (Fig. 2.12.4).

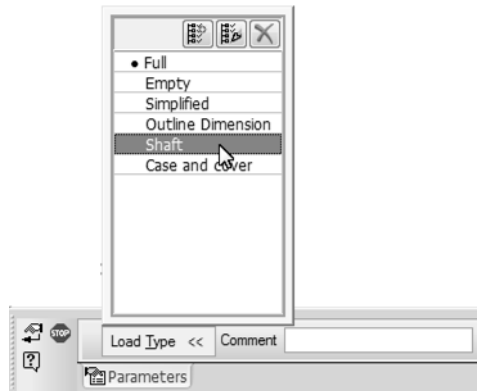


Fig. 2.12.4. Property bar in the process of working with loading types

Assembly loading types are displayed in the **Loading type** list. The applied load mode is marked with a circle or asterisk. The asterisk (*) instead of the marker in the list or in the **Applied loading type** field on the **Current state** bar means that the current set of component loading types differs from the applied set.

Creation of a loading type

To create a customized load mode for the current assembly, complete the following steps.

1. Preset the required loading types for assembly components (see Section 2.12.2.1.6 on p. 733).
2. You can prohibit editing of some components (see Section 2.12.3.1 on p. 744).
3. Call the command for working with the loading types.



Component loading and unloading, applying or removal of the read-only status are possible after calling the command.



4. Click the **Create** button above the **Loading type** list on the Property Bar.

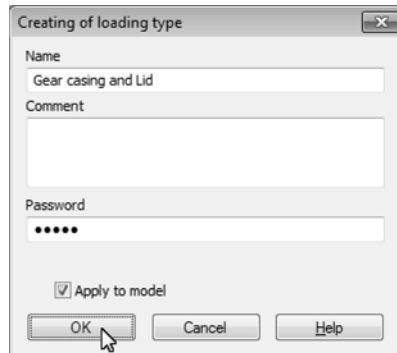


Fig. 2.12.5. Creation of a Loading Type for the Current Assembly

5. Enter the loading type name in the **Name** field of the **Loading type creation dialog**. The name should be different from other loading types names in the current assembly.



Names of the loading types created in subassembly component files can be the same. The same names can be used to simultaneously load subassembly components in an assembly with the loading type required for each of them; the loading type is defined by the settings of the user loading type with the same name.

6. If a created loading type should be protected by a password, enter the password in the **Password** field and confirm it in the appearing dialog.
For more information on password protection of load modes refer to Section 2.12.2.3 on p. 741.
7. In order to apply a new loading to a model, enable the **Apply to model** option. If the option is disabled, the applied loading type will be the same as it was before the creation.
8. Close the dialog by click the **OK** button.
9. In order to apply a different loading type, select it and click the **Apply** button on the Special Control Panel.
10. Click the **Interrupt command** button to leave the command.



Deletion of the loading type

To delete the customized load mode of the current assembly, complete the following steps.

1. Call the command for working with the loading types.
2. In the **load mode** list of the Property Bar select the loading type to be deleted.
3. Click the **Delete object** button above the list.
The selected loading type shall disappear from the list.
When you delete the applied load mode, the preceding in the list load mode becomes current.
4. To leave the command, click the **Interrupt command** button on the Special Control Panel.





A load mode can be password-protected, and several protected load modes can make hierarchical “chains” (see Section 2.12.2.3). Deletion of protected load modes has the following peculiarities:

- ▼ you need to enter a password to delete a load mode that is not current or slave to it (the corresponding dialog will pop up after you click the **Delete** button),
- ▼ you cannot delete the current load mode included into a “chain” but not being the first in it (the **Delete object** button is not available).

2.12.2.2.3. Selecting Assembly Loading Type

You can select the assembly load mode:

- ▼ before opening its file;
- ▼ while working with assembly.

Use the **Load Mode** list to specify the assembly load mode before opening it. This list appears in the file open dialog when you select an assembly file (*.a3d).

The **Loading Type** list contains **Full**, **Empty**, **Simplified**, **Dimensions** loading options, as well as customized options (if any) (Fig. 2.12.6).



Fig. 2.12.6. Selecting a loading type while opening an assembly

Select the required load mode and click button **Open** in the Open File dialog. The assembly shall be opened. The loading type selected during opening will be **applied** to the assembly. It means that all assembly component will be assigned with the **Full**, **Empty**, **Simplified**, **Dimensions** loading type. If a user loading type is selected, each component will have its specific loading type.

In order to apply another assembly loading type while working with it, choose it from the **Loading type applied to assembly** list in the **Current state** bar (Fig. 2.12.7).

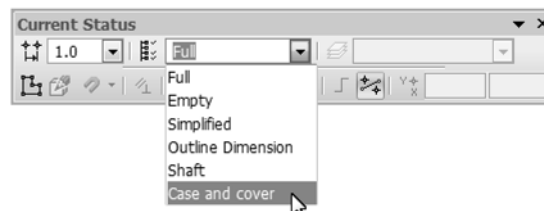


Fig. 2.12.7. Loading type selection for the current assembly on the **Current state** bar

Similar to assembly opening stage, this list contains names of the system and user (if any) loading types. You can create and apply a new type if it is missing in the assembly (see Section 2.12.2.2.2 on p. 736).

If a password-protected loading type is selected (see Section 2.12.2.3 on p. 741), the password entry dialog will appear on the screen except for the following cases:

- ▼ when a system loading type changes to another,
- ▼ when the selected load type is marked as applied with a circle or asterisk.
- ▼ when the selected load type is a subordinate one with respect to the applied mode.

Note that when loading an assembly file by specifying its name in the **File** menu, selection of load mode is impossible. An assembly will open with the load mode that was applied at the moment of its saving. The same is true when opening a subassembly of the current assembly with the **Edit in Window** command.

In order to change the subassembly component loading type to a type created in its source file, highlight the component in the Design Tree and click the **Loading type applied to assembly** button on the **Current state** bar. Specify the **Loading type** in the Property Bar and click the **Apply** button on the Special Control Panel.



2.12.2.2.4. Loading types editing

You can change the following during the loading type editing:

- ▼ in the system type — password and comment,
- ▼ in the user type — loading types of one or several components, loading type name, its password and commentary, as well as apply and remove read-only status.

To edit a loading mode of the current assembly, complete the following steps.

1. If you edit the state of a model, change loading of the components (see Section 2.12.2.1.6 on p. 733). If necessary, you can ban or allow editing of some components (see section 2.12.3 on p. 744).
2. Call the command for working with loading types in the same ways as it was done during their creation (see Section 2.12.2.2.2 on p. 736).



Component loading/unloading, applying/removal of the read-only status are possible after calling the command.

3. Specify the edited loading type in the **Loading type** list.
4. Click the **Edit** button above the **Loading type** list on the Property Bar.
5. Enter the password of the edited loading type, if it is password-protected.
6. Change parameters of the loading type in the **Loading type editing** dialog in the same way as it was done during its creation. For more information about password change, refer to Section 2.12.2.3.3 on p. 743.
7. In order to apply a new loading to a model, enable the **Apply to model** option. If the option is disabled, the applied loading type will be the same as it was before the editing. See Section 2.12.2.2.3 on p. 739 for information about application of loading types.



8. To transfer information about the current model's state (loading and read-only status of the model's components) to the edited type, enable the **Update to current model's state**. If the option is disabled, the information is not transferred.
9. Close the dialog by click the **OK** button.
10. To leave the command, click the **Interrupt command** button on the Special Control Panel.



2.12.2.3. Passwords for load modes

You can set passwords for system and custom assembly load modes.

A load mode that has a password is called a **protected load mode**.

Password is requested when you attempt to do the following:

1. opening a protected assembly, opening its subassemblies with protected loading type using the **Edit in window** or **Edit** in place command.
2. application of the password-protected load type.
3. deletion or editing of a protected loading type which is not applied,
4. removing read-only status from the component which was set as a result of protected loading type application (read-only status is indicated by a gray “shield” sign).

If incorrect password is entered, the above-mentioned operations will be blocked.



The password of system load modes can be used instead of the password of any custom load mode.

Password entry dialog is presented in Fig. 2.12.8.

Fig. 2.12.8. **Loading type password** query dialog

Loading type and **Access** fields are available only in case of an assembly opening.

From the **Loading type** list, you can choose another loading type. If a loading type unprotected by password is chosen, the **Password** field will be disabled.

From the **Access** list, you can choose a file opening variant for editing or for reading only.

You can set password on a loading type in the process of its creation or editing in KOMPAS-3D.



Password of the edited or applied loading type is not requested if a system loading type password was entered earlier to remove a component's read-only status (see Section 2.12.3.2 on p. 745).

2.12.2.3.1. Loading type password setting, editing and deletion

An existing password for system loading types is a prerequisite for setting password for the user loading type. In order to set, edit, or remove password, complete the following steps.

1. Call the command for working with loading types in the same ways as it was done during their creation (see Section 2.12.2.2.2 on p. 736).
2. Select the load mode to password-protect in the **Loading type** list on the Property Bar.
3. Click the **Edit** button above the list.
4. In order to set password, enter it in the **Password** of the appearing dialog.
A password may consist of 1 - 38 characters, including spaces.
Confirm password.
In order to change password, replace (*) symbols in the **Password** field with a new combination of symbols.
In order to delete password, remove all symbols from the **Password** field.
5. Close the password entry dialog with the **OK** button.
The protected load mode will be marked with a “key” in the list.



When setting a password for one of the system load modes, the same password will be automatically set for the rest system load modes.

6. If necessary, set a password for other load modes repeating steps 2-5.
7. To leave the command, click the **Interrupt command** button on the Special Control Panel.
8. Save the assembly file.



If the system load modes are protected, you can enter a password for the custom load mode directly in the process of its creation (see Section 2.12.2.2.2 on p. 736). Enter password in the **Password** field.



Password protection of all assembly load modes allows controlling access to nodes of a large assembly in case of the team work provided that different custom modes deny access to some components and allow access to others (see Appendix VII).

As a result of password setting for a user loading type, hierarchic “chains” can form (see Section 2.12.2.3.2).

2.12.2.3.2. Hierarchy of protected load types

Protected load modes can make hierarchical “chains”.

A “chain” forms if another protected load mode was **applied** at the moment of creating a new protected load mode (or at the moment of assigning a password to a custom load mode). In this case, the load mode being created (or protected) becomes **slave** with respect to the applied one, which in its turn, becomes **source** with respect to that being created (or protected). The list of load modes (when you work with the **Loading Types** command) will show the name of the slave load mode indented relative to the name of the source one (Fig. 2.12.9).

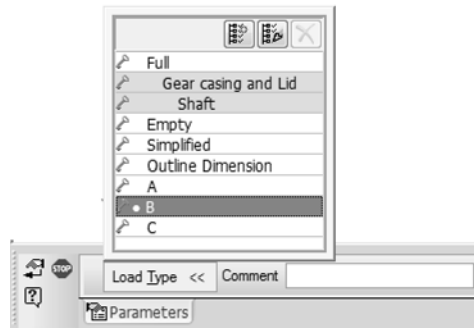


Fig. 2.12.9. Hierarchical “chains” of protected load modes

You can delete a slave load mode or its password only in case the source load mode is applied. Names of load modes that cannot be deleted or for which password cannot be disabled under the applied load mode are displayed in the list of load modes against the gray background (Fig. 2.12.9).



If a custom load mode is subordinate to the system one, it is sufficient to have any system load mode as the current mode to delete it.



If source load mode B with slave load mode C is, in its turn, slave to load mode A, then load mode A is considered source with respect to B and C; B and C are considered slave to A.

2.12.2.3.3. Loading type password editing and removal

For password removal, do what you did to set password for a model's loading type (see Section 2.12.2.3.1 on p. 742) but delete all symbols from the **Password** field in the **Edit loading type** dialog instead of entering symbols.

In order to change password, enter new symbols in the **Password** field and confirm the new password.



Editing and removal of the **applied** assembly loading type's password or any of its subordinate loading types do not require password entry.

When you delete passwords of the system load modes, a request will pop up to delete passwords of custom load modes. You can confirm or reject the deletion.

2.12.3. Component's read-only status

By default, all assembly's components can be edited. However, if necessary, you can set read-only status to a component.

The read-only status means that only a limited number of operations is available. For example, the component cannot be edited, moved, exploded; you cannot change its loading type and so on (see Section 2.12.3.3 on p. 746 for specific features of such components).

You can prohibit editing:

- ▼ “manually” for each component separately using special commands (see Section 2.12.3.1);
- ▼ by application of a loading type containing information about the read-only status (for information about loading type selection, refer to Section 2.12.2.2.3 on p. 739).

When working with an assembly, you can apply or remove the read-only status to components without limitations. Information about read-only status of components is not saved in system loading types.

The model's state taking account of components' read-only status can be saved in the user loading type only in the process of its creation or update (see Section 2.12.2.2.4 on p. 740).



Components added to an assembly after application of a password-protected loading type will automatically have the read-only status in other loading types.

2.12.3.1. Read-only status setting

You can apply or remove read-only status to/from one or several components on any nesting level of an assembly.

In order to apply or remove the read-only status for a component (or components), call the following command:

- ▼ **Editing — Prohibit** and **Editing — Allow** commands from their context menus (Fig. 2.12.10, a);
- ▼ **Read-only status for selected components** by clicking the same-named button on the **Assembly editing** bar.

After you call the **Editing — Read-Only** command, the component is assigned 'read-only' attribute and is marked with a “shield” in the Design Tree (Fig.2.12.10, b). For more details about using these components, see Section 2.12.3.3 on p. 746.

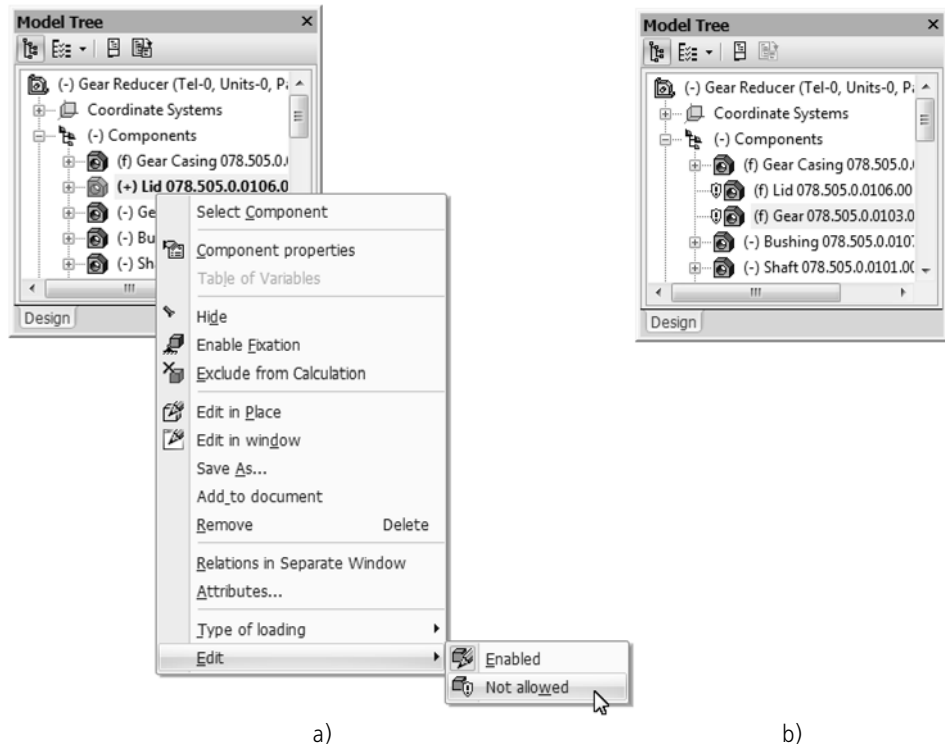


Fig. 2.12.10. Read-only status of a component a) setting, b) display in the Design Tree

In order to remove the read-only status from the selected component(s), the following command should be called from its context menu: **Editing — Allowed** or click the **Read-only status for selected components** on the **Assembly editing** bar. The “shield” next to the component in the Design Tree disappears.



You can create a new loading type by setting the read-only status or edit an existing loading type. For more information about saving the read-only status, refer to Section 2.12.3.2 on p. 745.

2.12.3.2. Saving read-only status for components

Information about the component's read-only status can be saved in the user assembly loading type. It is saved at the moment of an existing loading type creation or update. For more information about loading type creation, refer to Section 2.12.2.2.2 on p. 736; see section 2.12.2.2.4 on p. 740 for more details about editing.

The component's read-only status will not be saved in the assembly unless it is saved in the loading type.

After application of the selected loading type with information about the read-only status of its components, the respective indications will appear in the Design Tree.



- ▼ blue “shield” signs — if loading type is applied which is not password-protected,



- ▼ gray “shield” signs — if a password-protected loading type is applied (see Section 2.12.2.3 on p. 741 for password setting).

The user can remove the read-only status from the selected component using the **Editing — Allowed** command, for example (see Section 2.12.3.1 on p. 744). If a component is marked with the gray “shield” sign, you will have to enter the system loading type password to remove the read-only status.



Removal of the loading type password **does not remove read-only status** from components automatically — the “shield” sign will remain gray.

In order to allow editing of such a component, call the **Editing — Allowed** command after password removal.

In order to save the read-only status in a loading type created earlier, perform the following operations:

1. Apply the loading type to which the read-only status should be assigned. For example, choose it from the **Loading type applied to assembly** list on the **Current state** bar (see Section 2.12.2.2.3 on p. 739).
2. Apply the read-only status to components in the model.
3. Call the command for working with the loading types (see Section 2.12.2.2.2 on p. 736).



Applying/removal of the read-only status is possible after calling the command.



4. Call the **Edit** command by clicking the button above the list. Enable the **Update to current model's state** option in the dialog. Close the dialog by click the **OK** button.
5. Save the assembly.



It is impossible to assign the read-only status to the system loading type. You can create a user type and apply the read-only status during system loading.

So, using protected load modes you can control access of different users to various assembly components during team work.

2.12.3.3. Specifics of working with read-only components

This section lists restrictions imposed on working with read-only assembly components.

2.12.3.3.1. General specifics

If a component has the read-only status, you cannot do the following when working with it:

- ▼ delete it from an assembly in any way,
- ▼ edit it in a window or in the assembly context,
- ▼ control inclusion in calculation,

- ▼ edit properties,
- ▼ explode it (if a component is a subassembly),
- ▼ move and rotate it,
- ▼ change a load mode.

2.12.3.3.2. Working in the model construction tree

If a component is read-only, the following restrictions apply when you work with it in the Design Tree.

- ▼ The commands of the context menu that cannot be executed are inaccessible.
After you call the **Properties** command, the Property Bar will have the **MP Parameters** and **Source File** tabs only. Tab controls are inaccessible.
- ▼ A component will not be included into a new subassembly obtained by merging selected components.
- ▼ You cannot view component composition.
- ▼ You cannot edit component name.
- ▼ Other components cannot be copied or transferred to a read-only subassembly.

2.12.3.3.3. Performing operations and building objects

- ▼ If a component is read-only, you cannot apply to it the operations that result in creation or deletion of faces and edges (**Chamfer**, **Hole**, **Array**, **Surface Splitting**, **Delete Faces**, etc).
If a component is provided with the “read-only” attribute in the result of some operation applied to it, this operation will be **blocked**. You cannot do the following if the operation is blocked:
 - ▼ edit,
 - ▼ directly change variable values (variables can be only assigned values by link or in the result of calculation of expressions),
 - ▼ delete,
 - ▼ control inclusion in calculation,
 - ▼ use as a source array element.

Blocked operations are marked with a “lock” icon in the Design Tree (Fig. 2.12.11). Blocked operations remain effective for read-only components.

After you disable the “read-only” attribute of the component, the operation applied to it will be unblocked.

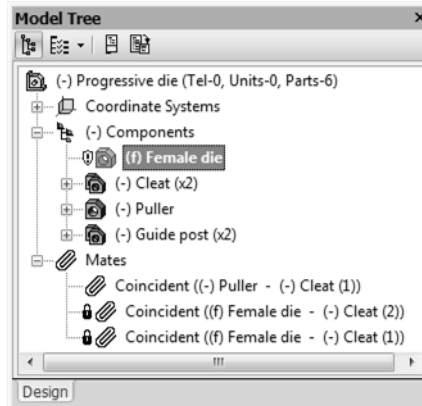


Fig. 2.12.11. Model objects blocked in the result of applying the read-only status to one of the components

- ▼ The commands not supposed to change faces and edges can use read-only components. For instance, a part face can be a draft base in the **Draft** command or the source surface in the **Thicken** command.

In addition, the objects belonging to a read-only component can be sources for new objects. For instance, a part face can be used to create a sketch or to place a symbol; points created in a component can be used to build auxiliary axes, planes, etc.



You can use a read-only component to build other objects if this component has the **Full** load mode.

2.12.3.3.4. Arrays of components and insertions from one source

- ▼ It is impossible to create an array of read-only components.
- ▼ If at least one of the source components of the created array is assigned the “read-only” attribute, this array and its instances will be **blocked** (similar to the operation applied to a read-only component, see above).
- ▼ The component copies, obtained both by array creation and by repeated insertion, make a group in the Design Tree. In the first case, all group components will be automatically assigned or unassigned the “read-only” attribute once it is assigned or unassigned to one of the group components. In the second case, the “read-only” attribute must be assigned or unassigned to each group component individually.



Change in a load mode for one component applies to all its insertions irrespective of whether they are readable or editable.

2.12.3.3.5. Mates

- ▼ A read-only component cannot be involved in mates.

- ▼ If at least one of the components involved in the created mate is assigned the “read-only” attribute, this mate will be **blocked** (similar to the operation applied to a read-only component, see above).
A blocked mate will not work.
- ▼ When you delete the assembly component with which a read-only component is mated, the blocked mate will be also deleted. After removal of this attribute or after you re-open the assembly, the component location will be recalculated in compliance with the rest mates.



When you delete an object used to mate (e.g. edges), the blocked mate will not be deleted but will be marked as failed.

2.12.3.3.6. Variables

- ▼ You cannot directly change variable values of a read-only component.
- ▼ Variables of a read-only component can be assigned values by link or in the result of calculation of expressions they are involved in.

2.12.3.3.7. BOM items

- ▼ You cannot edit a BOM item of a read-only component.
- ▼ Changes in a BOM item of a read-only component are transferred to an assembly after you remove this attribute from the component or after you re-open the assembly.

2.12.3.3.8. Rebuilding

- ▼ If the source file of a read-only component has been edited, it will be rebuilt automatically upon opening an assembly, without a request to rebuild.
- ▼ A read-only component is rebuilt in the ordinary way after you change its source objects or variable values by clicking the **Rebuild** button.

2.12.3.3.9. Model variants

The read-only status of the component is transferred from the source one to dependent ones, i.e. if a source variant component has the read-only status, editing of the respective component of the dependent variant will also be impossible. The “lock” sign next to the component's symbol in the Design Tree of the dependent version will be replaced by the “shield” sign.

It is impossible to cancel the link between such component and the source variant component, since the **Cancel link** command of the component's context menu in the Tree of Variants will be unavailable.

2.13. Model versions

2.13.1. General Information about Versions

2.13.1.1. Basic concepts

A model document can contain data about several model construction versions with common constructive attributes but having some differences. These variants are called **model versions** (hereafter — versions).

All model versions are contained in a single **group** document as individual models.

The **main** version is created by default during creation of the model. This version is the source variant of the model and is used for creation of other versions of the model.

You can create a version directly in the main version or use another version previously created on the basis of the main one. The version that is used for creation of other version is the **source** one.

The system supports creating **dependent** and **independent** versions. The dependent version is linked to the source version, while the independent one has no such link. Presence of such link is defined during creation of the version and cannot be changed later, i.e. it is impossible to make a dependent version independent and vice versa.

Objects of an independent version may be edited or deleted individually. They are not affected by changes in the respective objects of the source version.

Objects of a dependent version are linked to the respective objects of the source version. Changing or deleting objects of the source version is reflected in the dependent one.

A set of document's versions has a graphical representation in the form of **Tree of Versions** which is displayed in a separate tab of the **Model Tree** window. You can work with any version of the model after making it **active**. The models will be displayed as objects of the current version in the model window and in the model's Design Tree.

There are **permanent** and **variable** data in a model's version. Permanent data is identical for all model versions contained in a single document. Variable data differs for various versions.

2.13.1.2. Tree of Versions

The tree of the model's versions is displayed in the **Versions** tab of the **Model Tree** window (Fig. 2.13.1).

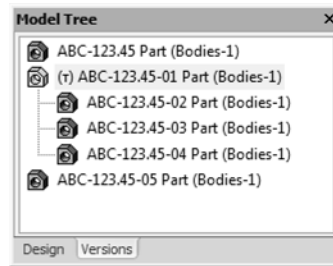


Fig. 2.13.1. Tree of model's versions

In the first position within the tree, there is the main version of the model; other versions, both dependent and independent, are displayed in other positions. All independent positions are located on the first level of the tree's hierarchy. Dependent versions are subordinate to the version which is the source version for them.

If necessary, you can hide dependent versions by double-clicking the icon of their source version. Double-clicking it again will display hidden versions.

Each version in the Tree of Versions has a name and an icon.

The name consists of a designation and a version's name. The designation is formed during creation of a version (see Section 2.13.2.2 on p. 757). The default name is the same as the name of the main version. Both designation and the name can be changed (see Section 2.13.3.4 on p. 763).



Note that, depending on the settings (see Section 9.2.7.4.13 on p. 1857), the version's name may contain a designation and a name or a name only. In the latter case, designation of the version is saved in the system but it is not displayed in the Tree of Versions.

Versions' icons are displayed in the Tree of Versions in blue color. If a version is the current one, its icon is yellow.



If a version is the dependent one, its objects are linked to the respective objects of the source version. A linked object in the Design tree is marked by the "lock" sign. It means that the object cannot be edited or deleted individually.

If necessary, you can create an additional window of the Design Tree for any version. For this purpose, choose the required version in the Tree of Versions and call the **Version in additional window** command in its context menu.

The additional window displays a copy of the Design Tree for the selected version. The header of the Tree's additional window contains the name of the version. Working with the additional window of the Design Tree is the same as that described in Section 2.1.2.4 on p. 109.

2.13.1.3. Versions in the Document Manager

In order to create a set of a model's versions, you can use the Document Manager (Fig. 2.13.2). You can use it to perform the following actions:

- ▼ create and delete model's versions, change their position in the list of versions,

- ▼ view properties of versions and change their values,
- ▼ set different values of variables for different versions,
- ▼ save data related to versions in files of different formats and use the saved data in order to create a set of versions in other models.

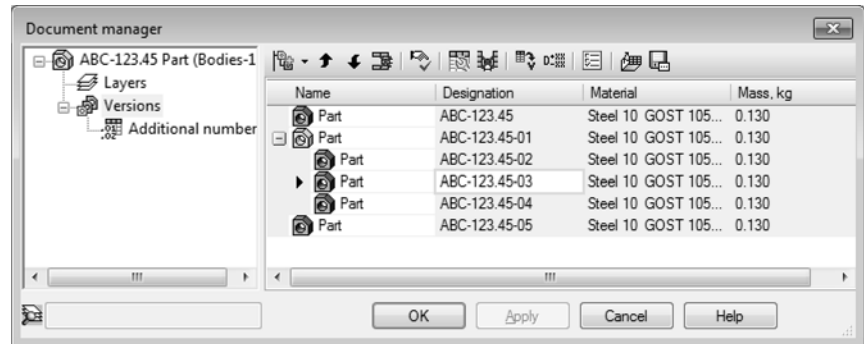


Fig. 2.13.2. Document Manager



The **Document Manager** dialog appears on the screen after calling the **Manage Versions...** command using one of the following methods:

- ▼ from the **Editor** menu,
- ▼ by clicking the button of the same name on the **Current status** bar,
- ▼ from the context menu of the version in the Tree of Versions,
- ▼ from the **Current Version** submenu of the root object's context menu in the Design Tree.



The first part of the dialog displays the document's Element Tree. If the dialog is called by the above-mentioned commands, the **Versions** element will be active in the Tree. The right section of the dialog displays the list of the model's versions created in the document and a toolbar for various operations with respect to versions.

The list of versions is displayed in the dialog as a table. Each column of the table corresponds to a certain property of the model.



You can create a set and specify order of properties in the table. For this purpose, click the **Customize Columns** button on the dialog's toolbar. The **Customize Columns** dialog will appear on the screen (working with the dialog is described in Section 5.1.4.2.2 on p. 1391).



Customization of columns with variables has certain peculiarities described in Section 2.13.3.5.2 on p. 766.



If necessary, you can disable display of any column, except for the **Name** column, without calling the customization dialog. For this purpose, activate the cell of the required column and click the **Delete Column** button.

In the Manager, you can add and remove a model's versions, change their position in a set of the document's versions, specify various values of properties for different versions, save information about versions and use this information for creation of a set of versions in other documents.

The **Document Manager** dialog makes it possible to work not only with model's versions but also with additional version numbers.



In order to perform different operation with respect to additional numbers, enable the **Additional Numbers** element in the dialog's tree. In the right section of the dialog, the list of additional numbers, created for a model with versions, will be displayed together with the toolbar for various operations to be performed with respect to additional numbers.

The list of additional numbers is displayed in the dialog as a table. Setup of the table's columns is the same as the setup of versions (see above).

In the Manager, you can add and delete additional numbers, change their position in the list, modify additional numbers and commentaries assigned to them, save information about the additional numbers and use this information for creation of additional numbers in other documents. Working with additional number is described in Section 2.13.3.6 on p. 770.

After performing all necessary operations in the dialog, close it by clicking the **OK** button. All changes will be saved. The **Cancel** button closes the dialog and without changes saving.

If you need to save changes without closing the dialog, click the **Apply** button.



Operations of adding, moving, and deleting variants will be saved automatically. It will not be possible to undo them in the Manager.

2.13.1.4. General procedure for working with versions

While working with versions, you are recommended to proceed as follows.

1. Create a new model. This model will be the main versions.
2. Specify a base designation and name of the main version (see Section 2.13.2.1 on p. 755). Each version created in the model will be assigned this designation and name.



Numbers of versions, created in the model are formed automatically in accordance with the settings. If default number forming parameters do not comply with the requirements adopted at your enterprise, change them prior to creating any versions. Setup of version numbers forming is described in Section 9.2.7.4.3 on p. 1845.

3. Create all objects and variables in the main version that should be transferred to other versions of the model.
4. Create the required versions of the model (see Section 2.13.2.2 on p. 757).



If necessary, you can change properties of the created version (see Section 2.13.3.4 on p. 763) or its position in the list of versions (see Section 2.13.2.3 on p. 760).

5. Edit versions' objects that should differ from the main version's objects. If necessary, you can create new objects in versions and delete the existing ones. Working with objects of dependent and independent versions is different (see Section 2.13.3.2 on p. 761).

Note that the model window and the model's Design Tree display objects of the version which is being current (active) at the moment. In order to edit a version, make it the current one (see

Section 2.13.3.1 on p. 761).



Changes made in the source version are transferred to dependent versions with some peculiarities described in Section 2.13.3.3 on p. 762.

6. Use variables to modify the geometry of dependent versions (see Section 2.13.3.5 on p. 765).
7. Edit the values of versions' properties that should differ from the values of the main version's properties (see Section 2.13.3.4 on p. 763).

You can insert the required model's version into another document (for example, into an assembly as a component), add an associative view to the drawing with geometry corresponding to the model's version, create a BOM for a model with versions, or create a table of the model's versions in a graphic or text document.



The sequence of creating versions and the order of their placement on the Versions Tree may be arbitrary. This does not affect their order of being listed in the BOM or table of versions.

2.13.1.5. Setup of versions

The following settings related to versions of a model are available in KOMPAS-3D:

- ▼ setup of creation of objects and variables in dependent versions (see Section 9.1.11.10 on p. 1735),
- ▼ setup of new version number forming in a new or current document (see Section 9.2.7.4.3 on p. 1845),
- ▼ setup of a BOM for an assembly with versions in a new document (see Section 9.2.5.2 on p. 1758).

2.13.2. Creation of versions

2.13.2.1. Designation and name of a version

The name of each model's version contains a designation and a name.

Designation of a version consists of a base designation and the version's number.

The base designation is common for all model's versions. The version number is formed individually for each version and has to be unique within the document.

The base designation is set manually. Version numbers are formed automatically but, if necessary, they can also be specified manually.

The number setting option (automatically or manually), as well as numbers' parameters that are formed automatically are defined during the setup process (see Section 9.2.7.4.3 on p. 1845). This setting is taken account of only when creating new versions and does not affect designations of versions that already exist in the model.

A version's designation can be specified:

- ▼ when working with the list of the version's properties in the Property bar (see Section 5.1.4.1.1 on p. 1385); this list is called by the **Version's Properties** command in the context menu of the Tree of Versions or the **Model Properties** command in the context menu of the root object in the Design Tree.
- ▼ when creating a new designation in the Tree of Versions (see Section 2.13.2.2.1 on p. 757),
- ▼ when working in the Document Manager (see Section 2.13.1.3 on p. 752) or in the Property Editor (see Section 5.1.4.2 on p. 1389).

In order to create a designation, a special dialog is used. In this dialog, you can specify the base designation and the version's number. Working with the dialog is described in Section 2.13.2.1.1 on p. 756.

The version's name can be specified when working with the list of version properties in the Property bar, Property Editor or in the Document Manager. The name's text is entered in the field of the respective property.

Later, it will be possible to edit designations and names of versions. The editing process is the same as the process of creation described above.

Methods of editing the versions' properties are described in Section 2.13.3.4 on p. 763; peculiarities of transferring the version's designation and name to other versions are described in Section 2.13.3.4.1 on p. 764.

2.13.2.1.1. Dialog for setting a version's designation

Designation of a version is specified in the **Designation** dialog (Fig. 2.13.3).

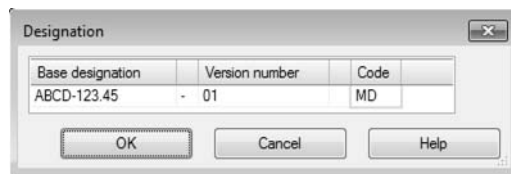


Fig. 2.13.3. **Designation** dialog

The **Base designation** field makes it possible to set the base designation of the version. Changes made in this field are transferred to designations of other versions of the model (for peculiarities of transfer of the changes, see Section 2.13.3.4.1).

In order to specify the version number, use the **Version number** field. Changing the number affects the current designation only.

The dialog also allows to change the default separator which is inserted between the designation's parts. As the separator, you can enter any symbol. This symbol will be used only in the current designation.



If the separator contained in a field is removed, and the field remains empty, the deleted separator will be restored in the designation once working in the dialog is finished.

Click the **OK** button to close the dialog. The **Cancel** button closes the dialog and without changes saving.

2.13.2.2. Method of version creation

You can create dependent and independent versions of a model using the Versions Tree or the Document Manager.

Using the Tree of Versions is preferred for creation of individual versions. It is recommended to use the Document Manager for creating a set of several versions.

The contents and the order of versions' positioning in the Manager is absolutely the same as in the Tree of Versions. When adding a version in the Manager, it will automatically added in the respective position of the Tree of Versions.

By default, the model document contains only one version – the main one. It is the source version for the first version to be created. Later, you can use not only the main version for creation of versions but also the versions created by the user.

When creating a version, its name and base designation are inherited from the source designation.



Specify the base value and the name of the main version prior to creation of all other model's versions (see Section 2.13.2.1).

The version number is formed automatically in accordance with the settings (see Section 9.2.7.4.3 on p. 1845) and is added to the version's designation.



If, during the setup process, you disable automatic generation of versions' numbers, the following peculiarities should be taken into account.

When creating a dependent version, its default name will be the same as the number of the source version. When creating an independent version, no default number will be generated. If you need the version's number to be unique within the document, in the first case, change the default number, and in the second case, specify the preferred number of the version.

On the basis of each model's version, you can create a dependent or independent version.

The created version, both dependent or independent, will inherit the geometry, layout elements, the set of variables and property values (including the color and optical properties) of the source version.



Objects that had been excluded from the calculation will be inherited with the *Excluded from calculation* attribute.

Procedure of creation of a version when working with the Tree of Versions can be found in Section 2.13.2.2.1, working in the Document Manager is described in Section 2.13.2.2.2.

2.13.2.2.1. Creation of a version in the Tree of Versions

1. Open the model's Versions Tree.
2. Specify the version which will be the source one for the version being created and call the **Create Version** command from its context menu. Control elements will appear on the Properties



Bar (Fig. 2.13.4).

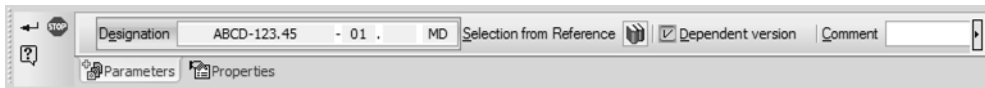


Fig. 2.13.4. Elements of creating a version

3. In the **Designation** field, the version's default designation is displayed. It consists of the base designation and the version's number.

You can modify any part of the created version's designation using a special dialog (see Section 2.13.2.1.1 on p. 756). In order to call the dialog, click the **Designation** field.

Note the following specifics when editing a designation:

- ▼ changing the base designation will be reflected in other versions of the model (see Section 2.13.3.4 on p. 763),
- ▼ the version's number has to be unique within the document; if numbers of the version being created and of a version that exists in the document are the same, the new version will not be created.



You can select a designation of the version from an external reference (Classifier of the Unified system for design documentation). To do that, click the **Select from Reference** button. The reference must be connected to KOMPAS-3D.

4. Check status of the **Dependent version** option. In order to create a dependent version, the option should be enabled; if you need to create an independent version, the option should be disabled.
5. If necessary, enter commentaries to the version being created in the **Commentary** field.
6. In the **Properties** tab, you can specify the color and optical properties of the version being created.



7. In order to finish creating the version, click the **Create Object** button. The new version will appear in the Tree. It will automatically be the current one.
8. In the same way, create all model's versions you need.



You can also create a version using the Design Tree of the current version. For this purpose, call the **Versions — Create Version** command from context menu of the Tree's root element. Further steps are the same as those described above.

2.13.2.2.2. Creation of a version in the Document Manager



1. Open the Document Manager (see Section 2.13.1.3 on p. 752). The **Versions** element will be enabled in the Manager's tree.
2. Choose the version, that should become the source one for the version being created, in the list of versions in the right part of the Manager's window.



3. In order to create a dependent version, click the **Create dependent version** button in the Manager's toolbar.



If you need to create an independent version, call the **Create independent version** command from the menu of the **Create dependent version** button.

The new version will appear in the list of version of the Manager and in the Tree of Versions. Properties of thee version, including its name and designation, will be displayed in the Document Manager.

If necessary, you can change the designation, name, or any other property of the version (see Section 2.13.3.4 on p. 763). You can also change the version's position in the list of versions (see Section 2.13.2.3 on p. 760).

4. In the same way, create all model's versions you need.



In order to create the list of versions, you can use the list of versions of a different model (see Section 2.13.2.2.3 on p. 759).

5. Click the **OK** button to close the Document Manager.

2.13.2.2.3. Using the list of versions of a different model

The set of versions of an existing model can be used for creating a similar set of versions in a different model. For this, do the following.

1. Open the Document Manager in an existing model with versions (see Section 2.13.1.3 on p. 752).



2. Click the **Save as...** button on the Manager's toolbar.
3. In the appearing file save dialog, specify the folder for saving, choose the **ODF Electronic Tables (*.ods)** or the **Microsoft Excel Electronic Tables (*.xls)** file type and click **Save**.

Information about the model's versions will be saved in the file of the specified type.

4. Open the model in which a version should be created.

5. Open the Document manager.



6. Click the **Read from file** button on the Manager's toolbar.
7. In the appearing file open dialog, choose file of the *ods* or *xls* format and click the **Open** button. The list of variants will be transferred to the model from the file.

Transferred versions will be added to versions existing in the model and positioned after them in the list in the Document Manager or in the Tree of Versions.

You can add versions from several files to the model.

Further operations with added versions is the same as working with the versions created in the model.

When creating a set of versions using the above method, not the following peculiarities.

- ▼ The base designation of versions added from file will not be transferred to versions that already exist in the model.
- ▼ All versions contained in the selected file are added to the model irrespective of whether the model has versions with the same numbers. Numbers of added versions do not change.

- ▼ Only the values of those versions' properties are transferred to the model that are not linked to the model's geometry. Geometry and its related properties, such as mass, properties created from variables of operations' parameters, and so on, cannot be transferred this way.

2.13.2.3. Changing a version's position in the list of versions



When working in the Document Manager, you can change a version's position in the model's list of versions.



For this purpose, choose the version you want to move and click **Move line up** or **Move line down**. The version will be moved, respectively, one position up or down in the list of versions in the Manager or in the Tree of Versions.

Availability of buttons depends on whether it is possible to move a version to a certain position. Versions can be moved within one level of the list.



Changing the version's position in the list does not change its number.

2.13.2.4. Removal of a version

You can remove a version in the following way:

- ▼ in the Tree of Versions, using the **Remove** command which is called from the **Editor** menu or from the version's context menu,



- ▼ in the Document Manager, by clicking the **Delete line** button in the Manager's toolbar.

You can delete any version, except for the main one.

If the deleted version has dependent versions, these versions will also be deleted.

If a dependent version, being the current one, is removed, the current status will be assigned to its source version. If an independent version is the current one, the main model's version will become the current one after its removal.

If necessary, you can delete several versions at once selecting them with the *<Shift>* or *<Ctrl>* keys.

2.13.3. Working with versions

In order to edit objects in a model with versions, the same methods are used as for models without versions.


Changes are made in the current version of the model. If the version has dependent versions, the new changes will be reflected in those versions. If the version is dependent, its objects will be linked to objects of the source version and will not be available for editing. If necessary, you can cancel links between objects and make the changes you need. For more information about version editing, see Section 2.13.3.2 on p. 761.

Parameters of versions' objects can be changed using variables. For more details about peculiarities of working with variables in versions, see Section 2.13.3.5 on p. 765.

A version created in a model can be used in other documents; for example, you can insert it in an assembly as a component or use it for creation of an associative view in a drawing. The required version is selected when operation is being performed. For example, when inserting a component into an assembly, version is selected from the drop-down list in the file open dialog.

2.13.3.1. Choosing the current version

Immediately after creation of a version in the Tree of Versions, it will become the current one. In order to manually make a version the current one, use one of the following methods:

- ▼ in the Tree of Versions, call the **Current** command from the context menu or click the version's name,
- ▼ in the Design Tree, call the **Current version** command from the context menu of the Tree's root element and specify the required version in the appearing list.
-  ▼ in the Document Manager, click the **Make current** button on the Manager's toolbar or call the **Current** command from the version's context menu,
- ▼ on the **Current status** panel (if it is positioned horizontally) — select the desired version from the list in the **Version management** field or type the version's designation in the field.

When opening a model with versions, you can specify the version to be the current one. For this purpose, choose the model's version you need in the **Version** drop-down list.

Objects of the current version are displayed in the model window and in the model's Design Tree.

2.13.3.2. Creation, editing and removal of a version's objects

A version created in a model inherits geometry from the source version. Later on, you will be able to change the version's geometry by adding, editing or removing its objects.

In order to display all objects in the model window or in the model's Design Tree, make the version the current one, as it is described in Section 2.13.3.1.



For convenient handling of a version, you can create an additional Design Tree for it. For this purpose, choose the required version in the Tree of Versions and call the **Version in additional window** command in its context menu.

An independent version is fully autonomous. It is not affected by changes in other model's versions.

Changes made in the source version are reflected in the dependent version.

Objects can be created in any version, both dependent or independent.

If a version, in which an object is created, has dependent versions, the identical object can be created in those versions. The object will be created, if the **Create object in dependent versions** option is enabled in the version setup dialog (see Section 9.1.11.10 on p. 1735). This option also controls adding components to dependent components' versions that are identical to those added to the source version.

Editing and removal of versions' objects have the following specifics.

Objects are available for changing and removal in independent versions.

In dependent versions, individual editing or removal of objects is not available (such objects are marked with the “lock” sign in the Design Tree).

If you need to change or remove an object of a dependent version, cancel its link to the object of the source version. For this purpose, call the **Cancel link** command from the object's context menu. The object will become available for editing and removal, and the **Restore link** command will appear in the context menu. This command makes it possible to restore the link between objects.



Once the link is restored, and the model is rebuilt, the dependent version's object becomes identical to the existing object of the source version. Changes made to the object of the dependent version prior to restoring the link will not be preserved.



Editing and removal of the source version's objects are reflected in dependent versions if the respective objects of dependent versions retain the link to objects of the source version.

In order to transfer changes of objects, rebuild the model with versions by calling the **View — Rebuild** command. Once the command is called, all versions for which rebuilding is required (marked with the red checkmark in the Tree of Versions) will be rebuilt. Deletion of objects is automatically transferred.

If necessary, you can change objects of the model's versions using variables. These operations are described in Section 2.13.3.5.

2.13.3.3. Transfer of objects to versions

Objects that are present in the current version of a model can be transferred to other versions, both dependent and independent. In order to transfer objects, do the following.

1. Make the version, whose objects should be transferred, the current one (see Section 2.13.3.1 on p. 761).
2. Call the **Editor — Transfer to versions** command. Control elements for joining curves will appear on the Properties Bar (Fig.2.13.5).

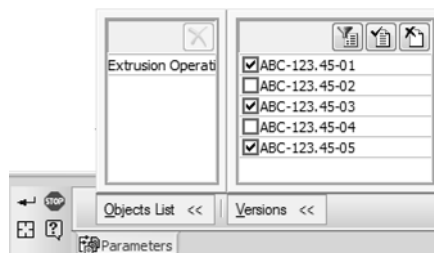


Fig. 2.13.5. Property bar elements

3. Create the list of the current version's objects that should be transferred.

In order to add the required objects to the list, select them by the mouse in the Design Tree or in the model window. The names of the selected objects will appear in the **List of Objects** toolbar.



If necessary, you can delete an object from the list of transferred objects by selecting it in the **List of Objects** bar and clicking the **Delete from list** button.

4. Customize the list of versions to which the selected objects should be transferred. For this purpose, use the **Versions** bar.



By default, this bar displays all versions that exist in the model. If you want the list to contain only dependent versions of the current version (versions that are subordinate to the current version in the Tree of Versions), click the **Only dependent versions** button.



The enabled option is displayed in the left of each version in the list. It means that the selected objects will be transferred to all listed versions. If you do not need to transfer objects to a version, disable its respective option.



In order to enable and disable all options in the list, you can use **Enable all** and **Disable all** buttons.



5. In order to transfer objects, click the **Create Object** button.



If the object being transferred has source objects, such as sketches, the objects will also be passed to the selected versions.



If the object being transferred is already present in the selected versions, this object will not be passed to these versions.

2.13.3.4. Editing properties of a version

A set of a model's properties is the same for all versions contained in it. Working with the set of properties is the same in models with and without versions (see Section 5.1 on p. 1369).

A peculiarity of models with versions is that the common set of the model's properties can be edited when working with any of its versions.

Property values may vary for different versions of the model.

When a new version is created, all values of the source version's properties are transferred to it. The only exception is the *Designation* property. Only part of the version's designation (the base designation) is transferred to the new version. The version's number is formed by the system or is set by the user.

Values of the version's properties can be edited when working with the list of the version's properties in the Property bar.

In the Tree of Versions, the list of properties is called for each of the model's versions by the **Version's properties** context menu command.

In the Design Tree, the list of properties is called for the current model's version. For this purpose, call the **Model properties** command from the context menu of the Tree's root object.

You can change values of properties for a version in the same way as for a model. The general procedure of handling the list of properties can be found in Section 5.1.4.1 on p. 1385.

Besides that, property values of the current version can be edited in the Property Editor (see Sections 5.1.4.2 on p. 1389 and 5.1.4.4 on p. 1395).

If you need to change values of properties for several versions, use the Document Manager (see Section 2.13.1.3 on p. 752). You can customize displaying of columns with all the required properties and specify values of different versions' properties.

Values of properties in the Document Manager are changed in the cells of these properties. The following properties are exceptions to the rule:

- ▼ *Material, Density, Mass* – these properties are edited in the **Mass center characteristics** dialog (see Section 5.1.4.4.2 on p. 1395),
- ▼ *Position, BOM section* – these properties are edited in the **Choose section** dialog (see Section 6.2.2.3.1 on p. 1472),
- ▼ *Designation* – this property is edited in the **Designation** dialog (see Section 2.13.2.1.1),
- ▼ *Substitute material and Coating* — the values of the properties are selected from an external reference file (the file must be connected to the system KOMPAS-3D).

Dialogs for property editing are called by double-clicking the cell of the respective property or by clicking the ellipsis button in the right section of the cell.

Changes of properties made for one of the versions are transmitted to the other versions. This process has some peculiarities described in Section 2.13.3.4.1 on p. 764.

The Document Manager can contain properties created from variables. Peculiarities of editing these properties are described in Sections 2.13.3.5.2 on p. 766 and 2.13.3.5.3 on p. 767.



A version's name, aside from the above-mentioned methods, can also be modified in the Tree of Versions (see Section 2.1.2.5 on p. 111).

2.13.3.4.1. Some peculiarities of exchanging changed properties between versions

When editing version properties, note the following peculiarities in transmitting changes to other versions.

- ▼ The way base designation and name changes are transmitted depends on what version these changes have been made to. The following scenarios are possible:
 - ▼ for the main version, changes are transmitted to the all model versions that have the same base designations/names as the main one,
 - ▼ for a dependent version, changes are transmitted to its initial version and dependent ones if such are available (i.e. all the versions located on the same branch of the Versions Tree as the version being changed),
 - ▼ for an independent version, changes are transmitted to its dependent versions, if any.

Changes in the number contained in the version's designation are taken account of in case of automatic generation of numbers; the number of the newly created version will be next after the greatest number in the list of versions.

- ▼ For properties *Material, Density, Mass* (in case mass is entered manually), *Substitute material, Coating, Developed by, Checked by, Approved by, T. contr., N. contr.*, and the properties cre-

ated from variables, — version property changes are transmitted to the dependent versions that have the same values of respective properties.

- ▼ For the properties *Author* and *Organization* — version property changes are transmitted to all model versions.
- ▼ Changes in other properties of the versions are not transmitted to other versions.

2.13.3.5. Using variables in a version

In some cases, you may need to modify objects of dependent versions retaining their links to objects of source versions. For this purpose, variables are used.

Each geometric object of a model has a set of variables. These include variables of operations' parameters and variables of sketches' dimensions. These variables are used for model geometry handling.



From now on, we mean that variables of sketches' dimensions that are used for handling the geometry, correspond to controlling dimensions (see Sections 2.9.2.1 on p. 631 and 7.2.1.10 on p. 1606 for information about controlling dimensions).

When saving links between objects of a dependent version and objects of the source one, it is impossible to edit objects of the dependent version individually. Accordingly, it is impossible to change values of variables created for these objects in the dependent version.

In order to handle the geometry of dependent versions using variables, user variables should be created (see Section 7.1.2.1 on p. 1576) and matched against the respective parameter variables of operations and dimensions.



The set of user variables of the source version is transferred to versions created on its basis. For more details about peculiarities user variables creation in versions, see Section 2.13.3.5.1.

Variables can be matched in the Variables Window (see Section 7.1.1.3 on p. 1571). The matching means that a user variable is set as the value of an operation parameter variable or dimension variable, i.e. the name of the user variable is entered in the **Expression** cell of the operation parameter variable or dimension variable.

Later on, you can set values for user variables. These values will be transferred to matching variables of operations' parameters and sketch dimensions' variables. As a result, the version's geometry will be changed.



To make sure everything is correct, create and match user variables in the main version of the model before creating other versions. In this case, each version of the model will have the same set of variables.

In order to specify different values of a variable for different versions, use the Document Manager. For more details about working with variables in the Manager, see Section 2.13.3.5.2.

2.13.3.5.1. User variable in a version

When creating a version, all variables of the source version are transferred to it (including user-defined variables), i.e. a set of variables is created which is identical to the set of variables of the source version.

Note the following specifics of transferring user variables to versions:

- ▼ a dependent version – link to the respective variable of the source version becomes the value of the user variable; this link replaces the contents of the respective **Expression** cell of the Variables Window (if a property is created on the basis of the user variable, the cell's contents will be transferred without changes and the link will not be created);
- ▼ an independent version – the transferred user variable is fully identical to the respective variable of the source version; contents of the **Expression** cell of the Variables Window is transferred without changes.

Later on, you will be able to add the user variable to any version created in the model.

If the version, to which a variable is added, has dependent versions, the new variable will be transferred to all these variables, since the **Create variable in dependent versions** option is enabled in the version setup dialog by default (see Section 9.1.11.10 on p. 1735). If you do not need to transfer variables, disable this option.

In order to transfer a user variable to dependent versions, you can use the **Apply to dependent versions** command. This command is available in the context menu of the user variable in the version with dependent versions.

In the model's Variables Window, you can edit user variables of dependent and independent versions. Note that the list of variables displayed in the Variables Window is related to the current version (see Section 2.13.3.1 on p. 761).

You can change the contents of the **Expression** cell of the required variable. This will result in changing the variable's value. This change will be transferred to dependent versions.

If a variable of a dependent version should have another values (other than the value of the source version's variable), replace the link contained in the **Expression** cell of the variable by the value you need. Link between variables of a dependent and independent versions will be lost. If necessary, you can restore it by creating the link to the variable of the required version (see Section 7.1.3.5 on p. 1583).



Values of user variables can be changed in the Document Manager for different versions (see Section 2.13.3.5.2).

2.13.3.5.2. Specifying the values of user variables

Values of user variables are set using the Variables Window (see Section 7.1.1.3 on p. 1571) or the Document Manager (see Section 2.13.1.3 on p. 752).

The Variables Window is used for setting values of user variables in the current version (peculiarities of values setting are described in Section 2.13.3.5.1).



If a model contains properties created on the basis of user values, you can use the Property Editor (see Section 5.1.4.2 on p. 1389) in order to set values of variables in the current version.

In order to set values of user variables for different versions, open the Document Manager and perform the following operations.

1. Choose a user variable, the value of which should be set.



For this purpose, click the **Insert Variable** button on the Manager's toolbar. The **Variables** dialog with the list of variables for the current version will appear on the screen (see Fig. 7.1.7 on p. 1584).

If the required user variable is present in the list of variables of another model's version, choose designation of the version in the **Version** drop-down list.

Specify the required user variable in the list of variables and click **OK**. The **Variables** dialog will close. A column will appear in the Document Manager table. Its name contains the name of the selected variable. And the property will be automatically created in document on the basis of the selected variable (see Section 5.1.3.2.1 on p. 1383).



The property created on the basis of the variable will be included in the set of the document's properties. You can edit it in the Property bar (see Section 5.1.4.1 on p. 1385) or in the Property Editor (see Section 5.1.4.2 on p. 1389).

If the property based on the required variables is already created in the document, you will not need to use the variable selection dialog. Customize displaying the column using the values of the property, as it is described in Section 2.13.1.3 on p. 752.

2. Cells of the resulting column are available for both independent and dependent versions of the model. Enter the required values in the cells.

Note that a change in the value of a variable made for a version is transmitted to all of its dependent versions that have the same value of the said variable.

3. In order to apply and transmit changes to the model before you finish working with the Manager, click the **Apply** button.
4. Click the **OK** button to close the Document Manager.

2.13.3.5.3. Extra options for working with variables

In some cases, when working with versions, you may need to set the value of an operation's parameter variable or of a variable of the sketch's controlling dimension. You can set values of these variables for an independent version or for objects of a dependent version that have no link to the source version's objects (for example, the link was canceled or object was created in a dependent version).

If you need the variable's value to be set for one version only, make the version the current one (see Section 2.13.3.1 on p. 761) and set the value of the required variable in the Variables Window. The values of variables are set for a version in the same way as for a model (see Section 7.1.3 on p. 1578).

You can also create a property on the basis of the required variable of the current version (see Section 5.1.3.2.1 on p. 1383) and set its value using the list of properties in the Property bar (see Section 5.1.4.1 on p. 1385) or in the Property Editor (see Section 5.1.4.2 on p. 1389).



If necessary, you can create properties on the basis of any variables, except for functional and interval ones. Peculiarities of setting values for properties created on the basis of variables are described in Section 5.1.4.4.5 on p. 1399.

If you need to specify values of an operation parameter variable or a sketch controlling dimension's variable for several versions, open the Document Manager, add the required variable (see Section 2.13.3.5.2) and specify its values for versions.



Changing the value of the operation parameter variable (dimension's variable) for a version will be transferred to its dependent version, if the link between the respective parameters (dimensions) of these versions is retained.

In case of a variable of the sketch's controlling dimension or a variable of an operation's parameter expressed in linear or angular values, you can set the value and assign a tolerance for the value of the respective parameter or a dimension tolerance. Assigning tolerances is described in Section 2.13.3.5.4.

2.13.3.5.4. Viewing and assigning tolerances when working with variables.

When working with variables in the Document Manager (see Section 2.13.1.3 on p. 752), you can view individual tolerances assigned to variables of sketch controlling dimensions and to variables of operations' parameters whose values are expressed in linear and angular values (more information about individual tolerances can be found in Section 2.10.2 on p. 667).



If the column with values of the required variable is not displayed in the Manager, add it using the **Insert variable button** (see Section 2.13.3.5.2 on p. 766) or customize the column display parameter (see Section 2.13.1.3 on p. 752), if a property is created on the basis of this value in the document.



By default, assigned tolerances are not displayed in the cells of the Document Manager's table. In order to enable displaying individual tolerances, click the **Display individual tolerances button**. If the button is enabled, values of parameters (dimensions) are displayed in the cells, as well as classes of accuracy and maximum deviations of parameters (dimensions). To disable displaying the tolerances, disable the button.



If the **Display individual tolerances** button is enabled, cells of variables of sketches' controlling dimensions and of operations' parameters, whose values are expressed in linear and angular values, will not be available for manual entry.

The Document Manager makes it possible to assign tolerances for the above-mentioned dimensions and parameter values.

Tolerance can be assigned to objects of an independent model's version or to dependent version's objects that have no links to the objects of the source version.

In order to assign a tolerance, complete the following steps.

1. In the variable's column select the cell that corresponds to the required version.
2. Call the dialog for assigning tolerance using one of the following methods:
 - ▼ if the individual tolerance display mode is enabled in the Manager, click the ellipsis button in the right section of the selected cell or double-click it.
 - ▼ if the individual tolerance display mode is disabled, call the **Assign tolerance** command from the selected cell's context menu.

After calling the command, the **Value and tolerance** dialog appears on the screen (Fig. 2.13.6).

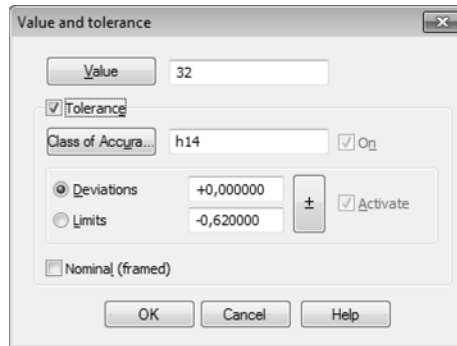


Fig. 2.13.6. The **Value and tolerance** dialog

3. In the appearing dialog, perform the required actions to set a tolerance. These operations are similar to those described in Section 2.10.2.2 on p. 669.

The dialog also makes it possible to set the value of the parameter or dimension to which the variable is related. It contains the **Value** field and the respective button (the button is present in the dialog for linear dimensions and parameters whose values are expressed in linear values).

Enter the value in the field or choose in the user menu that appears on the screen after clicking the **Value** button. By default, the menu contains normal linear dimensions according to GOST 6636-69.



If the value of a variable of a parameter or dimension is given in the form of an expression, you cannot change the value in the tolerance assignment dialog. The **Value** field and a same-name button are not displayed on the dialog.

4. Click the **OK** button to close the dialog.

The dialog will close. This value will be displayed in the Document Manager. If the **Display individual tolerances** button is enabled, the assigned tolerance will also be displayed in the Manager.



Values and tolerances specified for a version are transferred to all of its dependent versions after clicking the **Apply** or **OK** button in the Document Manager.

If you need to assign another tolerance to a parameter's value or dimension, cancel the link between the respective object of this version and the object of the source version.



The assigned tolerance is stored as maximum deviations of the parameter (dimension) in the Document Manager.

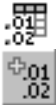
2.13.3.6. Additional numbers of versions

If a model's versions, aside from constructive differences, have such variable characteristics that may be shared by several versions (coating type, additional components of an item, and so on), additional numbers are created in the model.

Each additional number corresponds to a certain set of variable data. This data is set as values of additional numbers' attributes.

In order to create the list of additional numbers, perform the following steps.

1. Open the Document Manager (see Section 2.13.1.3 on p. 752).
2. In the Manager's tree, enable the **Additional numbers** element.
3. In the Manager's toolbar, click the **Create additional number of version button**.



The new additional number appears in the Manager. It will be formed in accordance with the settings (see Section 9.2.7.4.3 on p. 1845). If necessary, you can change the settings. These changes are taken into account only when creating new additional numbers.



4. Specify values of the additional number properties. For this purpose, make preliminary settings for the display of columns using **Column settings** and **Delete column** buttons (see Section 2.13.1.3 on p. 752). Values of properties are entered in the Manager's table cells.

If necessary, you can add columns with external variables and set values of the variables. Adding external variables and specifying their values are the same as operations described in Section 2.13.3.5.2 on p. 766.

5. In the same way, create all additional numbers you need and specify values of their properties. The list of additional numbers is displayed in the Manager as a table. The table cells are available for manual entry. If necessary, you can change the created additional numbers.

Besides that, you can change the order of the additional numbers in the list. Changing the additional number's position is described in Section 2.13.2.3 on p. 760.



In order to create additional numbers in the model, you can use another model's list of additional numbers saved in the *ods* or *xls* file. The numbers and values of their properties are added to the current model from the file.

If the model has a number identical to the number in the file (not only the numbers are the same, but values of their properties are also identical), the number will not be added to the model from the file.

Generating additional numbers using this method is identical to the procedure described in Section 2.13.2.2.3 on p. 759.

6. Click the **OK** button to close the Document Manager.

Variable data corresponding to additional numbers is presented as a table of additional numbers in a graphical or text document. In order to create the table, you can use the model or the associative view of the model created in a drawing.



The table of additional numbers is created by the **Create table of versions** command. The same set of controls is used for creation of both tables (versions and additional numbers). In order to create a table of additional numbers as a result of executing the command in a graphical or text document, select the **Table of additional numbers** table type in the dialog for table parameters setup (see Section 2.13.4.2 on p. 775).



All activities related to creation of the table of additional numbers are the same as activities related to creation of the table of versions (see Section 2.13.4.1 on p. 772).

Working with the created table of additional numbers is identical to working with the table of versions (see Section 2.13.4.3 on p. 776).

2.13.3.7. Saving a model with versions

Saving a model with versions is a process identical to saving a model without versions.

When saving a model with versions in KOMPAS-3D formats, information about all of its versions is saved.

When saving a model with versions in other formats, only information about the model's current version is saved.

You can also save any version in a separate file. For this, do the following.

1. In the Tree of Versions, call the **Create copy** command from the context menu of the required version. The **Information about document** dialog will appear on the screen.
2. Enter the required reference information in the dialog and click the **OK** button. Windows-standard File Save dialog will be displayed.
3. Specify the required name and location of the file on disk and click **Save**.
Only the selected version will be saved to the file. It will be the main one in the resulting file.

2.13.4. Table of versions

Information about versions' variable data is presented in the table of versions. In order to create the table of versions, you can use the model or the associative view of the model created in a drawing. The procedure of creating the table can be found in Section 2.13.4.1.

The table of versions can be located in a graphical or text document. You can edit data in the table, change its header, location and so on.

If the table of version is associatively linked to the source of data (a model or a model's associative view), you can retrieve data from the source document, open it, use commands for editing available for associative reports (see Section 5.2.3.8 on p. 1433). These operations are not available for the table which is not related to the source of data.

Working with the table of versions is described in Section 2.13.4.3 on p. 776.

Parameters of the table of versions are set during its creation or editing (see Section 2.13.4.2 on p. 775). Text of the table of versions located in a graphical or text document is displayed in accordance with the setup of text parameters of the report's table applied to that document (see Sections 9.2.4.7 on p. 1754 and 9.2.6.23 on p. 1835).

2.13.4.1. Creation of table

In order to create a table of versions, do the following.

1. Open a model or a drawing containing the model's associative view.
2. If the table is being created in a drawing, specify the required model's associative view. If the required view is not selected by the user, the current associative view will be automatically selected to create the table.



3. Call the **Create table** of versions command from the **Service — Reports** menu or using the respective button on the **Reports** bar.

Table creation elements will appear on the Property bar. They are located in two tabs: the **Table of versions** and **Components**.

Besides that, the **Parameters of table versions** dialog will be opened automatically for adjusting the table's appearance (see Section 2.13.4.2 on p. 775).

4. Customize the required settings in the **Parameters of table of versions** dialog and close it by clicking **OK**. Later on, you will be able to call the dialog by the **Setup...** button. Property Bars.
5. After calling the table creation command, the **Table of Versions** tab on the Property Bar will be activated (Fig. 2.13.7). Set the order of version designations' displaying in the table using the **Designation** switches on this tab.

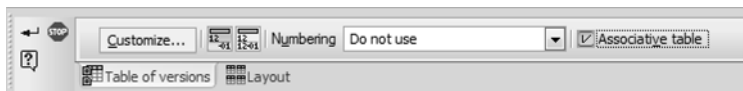
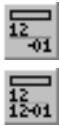


Fig. 2.13.7. Properties panel when creating a version table.
Table of versions tab.



By default, the **Only numbers of versions** switch is enabled. Full designation will be displayed for the first version in the table, while only numbers of designations will be displayed for other versions.

If you need the designation of each version to be displayed in full, enable the **Full designations** switch.

6. Check status of the **Associative table** option.

If the option is enabled, the table of versions will be associatively linked to the document being the source of data. If values of versions' properties will be changed in the source document, you will be able to reflect these changes in the table of versions (see Section 2.13.4.3 on p. 776).

If the option is disabled, the non-associative table of versions will be created. In this case, the data modified in the source document will not be transferred to the table.



If the table of versions is created for a model, the **Associative table** option will be available on the Property bar only if the model is saved to file on disk.



The associative table can be located in a graphical document only. If located in a text document, the table will become non-associative irrespective of whether the option is enabled or not.

7. You can enable automatic assignment of numbers to report tables. For this, select a numbering group on the **Numbering** list (about automatic numbering, see Section 3.9.1 on p. 1277).
8. When creating an associative version table for an associative view from a model in the drawing, you can link the table with the title block of the drawing (see Section 2.13.4.1.1 on p. 774).
9. If you need the list of the model's versions to be displayed in several tables rather than in one table, go to the **Components** tab (Fig. 2.13.8).

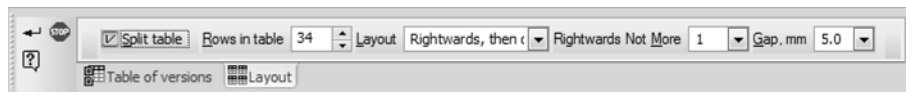


Fig. 2.13.8. **Components** tab

10. You can have several tables by enabling the **Split table** option.
11. In the **Number of rows** in table, specify the maximum number of rows in each table. In accordance with the value, the general list of versions will be divided into several lists; each of them will be placed in a different table. Display parameters are the same for all tables.
12. To specify the order of tables' positioning on the sheet, choose the required direction of positioning from the **Contents** drop-down list.

The **To the right, then downwards** option positions tables in horizontal rows. The number of tables in a row is specified in the **To the right, not more than** field.

The **Downwards, then to the right** option makes it possible to position tables in vertical rows. The number of tables in a row is specified in the **Downwards, not more than** field.

13. The **Gap, mm** field makes it possible to specify distance between the tables. You can enter the required value of the distance manually or select it from the drop-down list.



14. In order to finish creating the table, click the **Create Object** button.

If the table is being created for the model's associative view in a drawing, the drawing's field will display the table's ghost. Specify the location of the table's snap point using cursor or by entering coordinates in the Property Bar and click the left mouse button to fix the table. The table is placed in the associative view chosen for its creation.

If the table is being created in a model, the file open dialog will appear on the screen. Open the required graphical or text document. Locate the ghost of the table in the graphical document as described above. In a text document, the table is located automatically.



When creating the table of versions in a drawing, you can re-select the associative view for which the table is being created before the you finish working with the command. For this, all you need to do is click the desired view on the Tree or in the drawing window.

You can set a title for a table placed in the drawing, see Section 4.2.2.1.3 on p. 1364.

2.13.4.1.1. Link the drawing's title block to the associative table of versions

If there is a table of versions in the drawing with columns having *Material* and *Mass* properties, the cells of the drawing's title block that correspond to these properties should contain text indicating that the required data is contained in the table instead of the properties' values (for example, text as in *see Table*). This text is generated automatically.



- ▼ when the table of versions is created for the model's associative view (see Section 2.13.4.1 on p. 772),



- ▼ when the table of versions in a drawing is being edited by the **Edit table** command.

Text of the link to the table of versions can be generated only the table is an associative one (the **Associative table** option is enabled during creation of the table).

In order to create the link's text, the elements of the **Table of versions** tab in the Property bar are used (see Fig. 2.13.9).

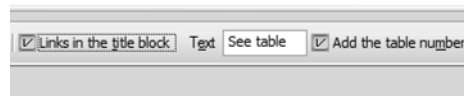


Fig. 2.13.9. Link formation elements

Links are created with the **Links in title block** option enabled. The field on the right of the option contains the default text of the link. If necessary, you can edit the text.

If the version table has a number automatically assigned to it (i.e. in the **Numbering** list, an item other than **Do Not Use** has been selected), the number must be added to the reference text. To do this, enable the **Add the table's number** option.



Only the number is added to the link's text; the text before and after the number which is assigned during the numbering parameters setup (see Section 9.2.6.22 on p. 1832) is not added.

Once the process of creation and editing of the table of versions is finished, contents of *Material* and *Mass* cells of the drawing's title block will be replaced by text of the link to the table. It happens if the table of versions contains columns of their respective properties. If a column of

a property, for example, the *Mass* property is not present in the table, the link's text will be displayed only in the *Material* cells, while the value of the respective property will be displayed in the *Mass* cell.

2.13.4.2. Setup of a table's parameters

The parameters of the table of versions are set up in the dialog illustrated in Fig. 2.13.10.

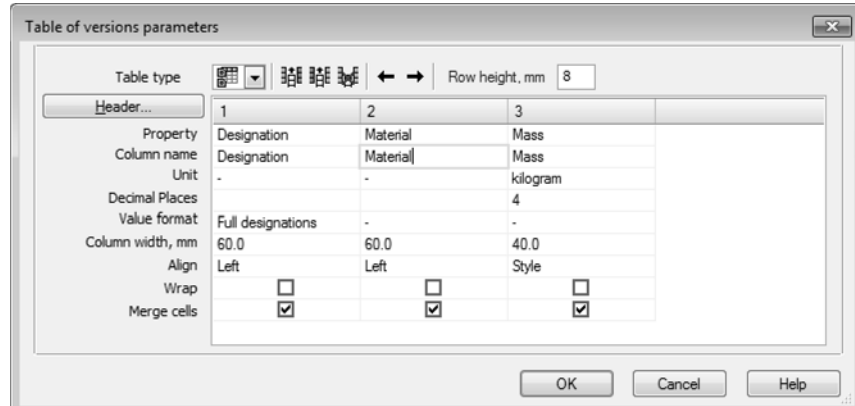


Fig. 2.13.10. The dialog for setup of the table of versions' parameters

The dialog is called when the table of versions is being created or edited by clicking the **Customize...** button on the **Table of versions** tab of the Property bar.



Besides that, the dialog appears automatically after the **Create table of versions** command is called.

The dialog contains a window for the table setup and a toolbar.

It is designed not only to set up parameters of the table of versions, but also for the table of additional numbers (see Section 2.13.3.6 on p. 770 for more information about additional numbers). The type of the table for setup is selected in the **Table type** drop-down list located in the dialog's toolbar. The set of the dialog's controls depends on the selected type.

As part of the table of versions setup, the following activities are available.

- ▼ Adding, deletion, and moving of the table's columns; controls are the same as those used for the report table setup (see Table 5.2.2 on p. 1408).
- ▼ Setup of the table's columns display parameters and data contained in them (see Section 2.13.4.2.1).
- ▼ Setting the table's row height. For this purpose, the **Row height, mm** field is used. The value is manually entered in this field.
- ▼ Setup of the table's header (see Section 5.2.2.2.4 on p. 1411).

In order to finish setup of the table's parameters, click **OK** in the **Parameters of table of versions** dialog.



If the list of versions is presented in several tables, each of them will be displayed in accordance with the setup. The created header will be contained in each of the tables.

2.13.4.2.1. Setup of displaying the table's columns

The following parameters can be adjusted in version table columns:

- ▼ **Property, Column name, Unit, Decimal points, Column width, mm, Alignment, Wrap**; the setup is identical to the setup of the report style parameters which is described in Table 5.2.3 on p. 1410,
- ▼ **Value format, Merge cells**; the setup is described in Table 2.13.1.

Tab. 2.13.1. Additional setup parameters of the table of versions

Element	Description
Value format	<p>This field* makes it possible to choose the variant of displaying a property's value in the table of versions. For example, the following display options are available for the <i>Designation</i> property.</p> <ul style="list-style-type: none"> ▼ <i>Full designations</i>, ▼ <i>Only version numbers</i>. <p>Choosing the required option for the property is the same as using the Designation switch in the Property bar.</p>
Merge cells	<p>This option makes it possible to merge adjacent cells in a column of the table of versions, if the cells' contents is identical, i.e. if the displayed property has the same value for adjacent versions.</p>

* The control allows to select an option from the drop-down list. After you click a cell, in its right corner there appears a button with a triangle. Press the button to display the list for selecting a parameter option.

2.13.4.3. Working with the table

The table of versions can be associatively linked or not linked to the source of data. The table's associativity is defined during its creation (see Section 2.13.4.1 on p. 772). If necessary, you can cancel the associative characteristic of a created table as is described in Section 5.2.3.8.3 on p. 1436, but it is impossible to make a non-associative table to be the associative one.

Typical commands are used for handling non-associative tables.

In case of associative tables, both typical commands (for example, cutting, copying, changing style) and special commands are available that are the same as commands for handling the associative report.

The associative table of versions is linked to the model or the model's associative view which was used for its creation. It is displayed in the same way as the associative report in an additional external dashed frame (see Fig. 5.2.14 on p. 1434).

If the associative table is divided into several tables, all of these tables are a single objects and will be displayed within a common frame.

You can update data in the associative table, open the source document for editing objects, change the table's snap point. These operations are similar to those described in Sections 5.2.3.8.1 on p. 1434 and 5.2.3.8.2 on p. 1435.



In order to edit an associative table, the **Edit table** context menu command is used (the menu is called for the highlighted table). The command can also be called by double-clicking the external frame of the table. After calling the command, table edit elements appear on the Property bar. When editing a table, you can perform the same operation as when creating it (see Section 2.13.4.1 on p. 772), except for enabling/disabling the associative attribute.



In order to finish creating the table of versions, click the **Create Object** button in the Property bar.

Apart from the above-mentioned activities, you can also specify the name of the associative table or tables into which it is divided. For this purpose, highlight the associative table and call the **Name...** command from the context menu. The dialog for table name entry will appear on the screen. Working with the dialog is described in Section 4.2.2.1.3 on p. 1364.

The data contained in the table, both associative and non-associative, can be edited in the document's field. In order to enable the edit mode, double-click the table (in case of the associative table, you can click anywhere, except for the dashed frame). This mode supports all tools for handling tables (see Section 4.2.1.1 on p. 1349).

2.13.5. BOM for assembly with versions

2.13.5.1. Creation of a BOM for assembly with versions

In order to create a BOM using an assembly with versions, you can use any of the following commands:



- ▼ **Create BOM for assembly** (see Section 6.2.3.8 on p. 1506).

This command generates internal BOM objects in the assembly and creates a BOM document connected with the assembly. BOM style is defined by the settings for new BOMs for assemblies, see Section 9.2.5.2.1 on p. 1758. By default, for assemblies with no more than 3 versions, the group BOM under A variant (GOST 2.113–75) is used, while for assemblies with more versions, BOM of the B variant is used (GOST 2.113–75, F.1, 1c). If necessary, you can change the default settings.



- ▼ **Create BOM objects** (see Section 6.2.3.7.1 on p. 1504).

This command forms BOM objects in assembly components, creates and saves the BOM document connected to the assembly (optionally). BOM style is defined by the general settings for new BOMs for assemblies, see Section 9.2.5.1.1 on p. 1755. By default, for new BOMs, the *Simple BOM (GOST 2.106–96)* style is used; therefore, before applying the **Create BOM objects** command to the assembly with versions, the setting should be changed by selecting one of the styles for group BOMs as the default one.

2.13.5.2. Components with identical position numbers

Sometimes it is required that certain components in BOMs for different assembly versions have the same position number. Normally, it is required for components that perform the same func-

tion but represented by different models in different assembly versions (or by different model versions).

To assign the same position number to some KOMPAS-3D assembly components, they have to be combined into a *group of components with the same position*. There can be several such groups in a single assembly. Once component may be included in one group only. Information about groups of components is stored in the assembly file.

All components included in a single group will have the same position number in BOM.



Groups are handled in the **Component groups** window. In order to enable/disable it, the respective button in the **BOM** bar is used. The **Component groups** window's interface and methods of working with it are described in Section 2.13.5.2.1.

Procedure for forming a group of components is described in Section 2.13.5.2.2.



If an assembly contains several identical components, it is not obligatory to include them in a single group. For example, only four out of ten bolts can be included in the group. As a result, these four bolts will have the same position number, while other bolts will have different position numbers which is defined by objects' sorting within BOM.

2.13.5.2.1. Component groups window

The **Groups of components** window (Fig. 2.13.11) is used for working with groups of components that must have the same position number in the BOM. You can enable and disable displaying the window by clicking the **Component groups** button in the **BOM** bar.

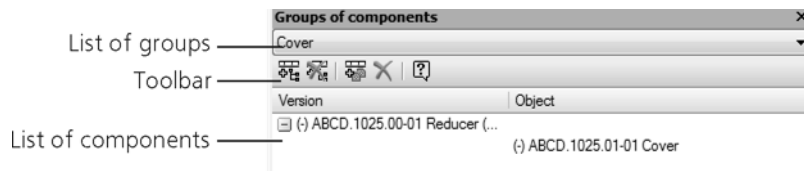


Fig. 2.13.11. **Component groups** window

The main part of the **Component groups** window is occupied by the list of components included in the current group. The list is represented as a table: in the first column, designations of assembly's versions are displayed; in the second column, designations of components included in these versions, the positions of which should be identical. Upon selecting a component in the list, it is highlighted in the Model window.

Above the list of components, there is the **Component groups** window's toolbar. Its buttons are only available when working with an assembly.

Above the toolbar, there is a field with the list for selection of the current group. Groups in the list are sorted in alphabetical order.

Methods of working with the component groups are described below.

In order to save the results of working with component groups to the model's file, the file should be saved. It is impossible to undo operations when working with groups; therefore, in order to cancel creation/deletion/modification of groups, the assembly file should be closed without saving.

Creation of a component group



In order to create a new group of components, click the **Create group** button on the toolbar of the **Component groups** window. A dialog for new group name entry will appear on the screen. Enter the group's name and click the **OK** button in the dialog. The dialog will be closed, the created group will be added to the list of groups; it will be automatically made the current one. The new group does not contain components, therefore, their list is empty.



It is impossible to rename group components.

Removal of a component group



In order to delete a group (or groups) of components, click the **Remove group** button on the toolbar of the **Component groups** window. The **Remove groups** dialog will be displayed (Fig. 2.13.12). Perform one of the following actions in it:

- ▼ in order to delete all groups, choose the **All** option,
- ▼ in order to delete one or several groups, choose the **Groups** variant and specify groups for deletion in the dialog's list.

After that, click the **OK** button in the dialog. By default, the **Groups** option is enabled in the dialog, and the group that was the current one when the dialog is called will be highlighted.



Fig. 2.13.12. **Delete groups** dialog

Removal of a component group from the assembly means deletion of information about this group from the file. Components included in the group will remain in the assembly in their positions.

Adding components to a group



In order to add components to the current group, select them in the Design Tree and click the **Add to group** button in the toolbar of the **Component groups** window. Designations of an assembly's version and of the selected components appear in the **Component groups** window's list.

If components being added are included in a different group, they will be automatically removed from this group.



As a rule, components of different assembly versions are included in the group; however, if necessary, you can create groups of a single version's components, as well as create component groups in assemblies without versions.

Removal of a component from group



In order to delete a component from the group, select it in the list of the **Component groups** window and click the **Delete** button on this window's toolbar.

Designation of the component will disappear from the list.

2.13.5.2.2. Forming a group of components with one position in an assembly

Forming a group of components in an assembly consists in creating a new (or selecting an existing one) group and including the required components from different versions.

In order to create component groups with the same position in an assembly, perform the following steps.



1. Enable displaying of the **Component groups** window.

2. Create a component group. To do this, complete the following steps:



2.1. Click the **Create group** button in the **Component groups** window's bar,

2.2. in the appearing dialog, enter the new group's name and close the dialog with the **OK** button.

3. Add components to the group. To do this, complete the following steps:

3.1. Make current that of the assembly's versions which contains components with the same position numbers as components in a different version.



3.2. select these components in the Design Tree and click the **Add to group** button in the toolbar of the **Component groups** window.

The selected components will appear in the table: in the **Version** column, the assembly's version designation is displayed; in the **Object** column, component designations are displayed (see Fig. 2.13.11).

3.3. Perform steps described in paragraphs 3.1 and 3.2 with respect to other components that should have the same position number as components that have already been added to the current group.



4. Call the **File — Save** command so that information about component groups is saved in the model file.



If the group, to which components should be added, already exists, activate the required group instead of creating the new one. For this purpose, choose its name from the list of groups.

2.14. Model Editing and Setting Up

2.14.1. Managing the state of objects Displaying

2.14.1.1. Enabling and disabling of the object visibility

The auxiliary (construction) axes and planes (especially when there are many of them in the model), the sketches, components and other objects not being used by any operations sometimes obstruct the model view. For a more convenient when working with the model you can make any of these items invisible. It will still be considered in hierarchy, and its derivative items will be displayed correctly.

You can manage visibility of the following objects:

- ▼ individual within an object (see 2.14.1.1.1),
- ▼ any auxiliary objects or groups of the auxiliary objects of the same type (see 2.14.1.1.2),
- ▼ any objects within a layer (see 2.14.1.1.3).

2.14.1.1.1. Managing visibility of individual components

To hide one or more items, select them in the Design Tree or in the model window and use the **Hide** command from the context menu. The objects will become invisible. Their respective icons will remain at their places in the Construction Tree, but will be displayed in gray color.

To make a or several object components, select them in the Model Tree and use the **Show** command from the context menu. The objects will become visible.

Please note that the body is always hidden and revealed completely even if a separate shape-forming operation was selected while calling the **Show** or **Hide** command. For instance, on executing the Show Boss-Extrude command all the bodies which contain this element will be displayed whereas Boss-Extrude Hide command will hide the entire body.

The same is true for surfaces. For instance, as the result of showing or hiding the surfaces which are involved in the **Knitting** operation, all other surfaces involved in this operation will be shown or hidden in the model window.

Upon calling the **Show** or **Hide** command for all other items (auxiliary axes, curves, points, etc.), only these items will be shown or hidden in the model window.



The **Hide (Show)** command is not available if all the selected objects are already hidden (shown).

Using the **Hide** or **Show** command for an object changes the *Visibility* attribute of this object to **Hidden** or **Visible** respectively. You can also edit this attribute manually without calling this command. To do this, use the **Properties** window, see Section 1.4.2.2 on p. 82.

2.14.1.1.2. Managing visibility of auxiliary components



To hide all the auxiliary objects in the model, click **Hide all objects** on the **View** panel. To hide the auxiliary objects of a certain type, call the corresponding command from the drop-down

menu of this button (Fig. 2.14.1). Upon calling any of those commands all the objects of the respective type, both existing in the model and the new ones, will be hidden. Hidden component icons are displayed in gray in the Design Tree. The button adjacent to the command name in the menu switched to pressed indicating that all the objects are hidden.

If there are any components in the model, you can customize visibility of each component. To do this, use the **Hide all objects in the components** button on the **View** panel. The button menu is identical to **Hide All Objects** button. The difference is that the **Hide all objects in components** command can be applied to the assembly components only whereas **Hide all objects** can be applied to both the assembly and its components.

To display the hidden auxiliary components, call the hide command once again.



The objects hidden using the above commands cannot be displayed by means of the **Show** command since it is not available in the context menu of these objects. You cannot change the *Visibility* attribute of these objects in the **Properties** window.

The menus of **Hide All Objects** and **Hide All Objects in Components** may be used for building separate instrument panels (see example in Fig. 2.14.2) to be used instead of menus. To do this use the mouse to drag the button menu by the header in the required direction.

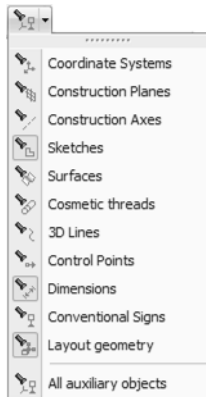


Fig. 2.14.1. Defining object type for hiding



Fig. 2.14.2. **Hide** Panel

Please note the difference between **Hide** and **Hide in Components** panels from the rest of the instrument panels: you cannot change the composition or the order of buttons on them.

2.14.1.1.3. Managing visibility of objects in a layer

You can manage visibility of the objects using the layers (for details about layers, see Section 2.15.13 on p. 845).

You can hide or display objects located in the same layer at once by editing the *Visibility* attribute of this layer.

Note that the *Visibility* attribute of each object must set to **By layer**, otherwise changing the layer's attribute will not affect the object's attribute.

The **By layer** is the default value of the *Visibility* attribute for all objects.

If the *Visibility* attribute of an object is not set to default value, that is, is set to **Hidden** (or **Visible**), then the object is hidden (or displayed) independently of whether its layer is visible or not.

To view or change the object attributes, use the **Properties** window, see Section 1.4.2.2 on p. 82.

2.14.1.2. Excluding Objects from Calculations

You can exclude from calculations any model features and components. When excluding an object from calculations, the model rebuilds so as if the specified object is deleted, however the information about it is still kept in the document.

To exclude one or several objects from calculations, select them in the Construction Tree and call the context menu command **Exclude from Calculation**.

After calling the command, the model will be rebuild, excepting the excluded objects and their dependent operations and objects. The icons of the objects excluded from calculations will be displayed in gray and marked with a 'cross' in the Construction Tree.



Objects excluded from calculation are temporarily removed from the model. Hence, the objects referencing them can sometimes be marked as failing objects. Errors disappear after including objects into calculations.

To re-enable the inclusion of one or several objects into calculation, select them in the Construction Tree and call the context menu command **Include into Calculation**.

Excluding objects from calculations helps to save calculation time when building additional elements or rebuilding the model, as well as to see how the model looked like before creating certain elements.

For example, after creating a sequence of dependent elements, you need to proceed to creating other elements that are not related and do not intersect with the existing ones. You can exclude the existing sequence from calculations, after which the model rebuilding time will significantly shorten when creating new elements (the system will not run checks for intersection on all elements and calculate the intersection lines). When the required elements are created, include the rest of elements into the calculations. Their relative position check will be run just once.

You can exclude from calculations chamfers, fillets, holes and other small elements, if, due to their large cumulative number, new geometry-producing feature creation is too slow.

Using **Exclude from Calculation** command, you can exclude objects from calculation selectively, i.e. regardless of their position in the hierarchy of model features and, accordingly, in the Construction Tree.



Remember to include in calculations all model elements when finishing a model development!

There can be examples not only of excluding features temporarily from calculation, but only of including features temporarily in the calculation.

Sometimes, when generating a flat model view (e.g. its isometric view in the drawing or a dimmed image in the folder), you will need to show a cut in the model (for example, a 1/4 cut

in the model). After the model has been created, do a trimming by sketch. Save the model file. Generate the desired model view. To continue working further with the model (for example, to calculate its mass-inertia properties or transfer the model to a process control sequence generating application), exclude the trimming operation from the calculation.

2.14.2. Color and properties control of item surfaces

You can also set the color and properties of the surface (flares, transparency, etc.) both for the entire model and its objects: components, bodies, surfaces, operations, faces, etc. For objects without surfaces, such as curves or points, you can only set the color.

To set a color and properties of the object's surface, select it in the Design Tree or in the model window and choose **Properties** from the context menu (**Model properties** for a model, or **Component properties** for a component).

To set the color and properties of the edge surface, you can use the **Face properties** command from the context menu of the edge currently selected in the model window.



If a part of an object is selected in the model window, e.g. an edge of an operation or a curve's segment, then the **Source item properties** command will be available in the context menu. Using this command, you can customize the properties of the parent object.

Once you call the command, the Properties panel will display the object properties controls, including the **Assign color** group of switches and the **Optical properties** panel with the sliders to adjust the optical properties of the surface (Fig. 2.14.3).

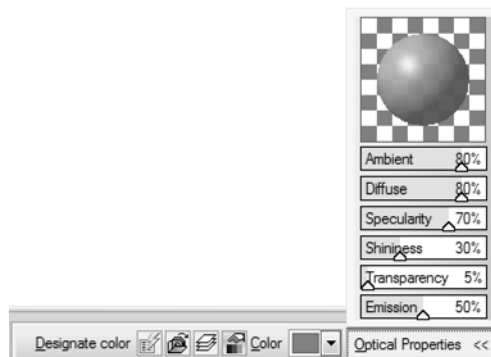






Fig. 2.14.3. Controls for adjusting the color and properties of the object surface

The available items in the **Assign color** group depend on the type of object you're configuring. The switches in this group are described in Table 2.14.3.

Tab. 2.14.1. Switches in the **Assign color** group

	Switch name; objects it is available for	Result of applying the switch
	By Source Object available for all objects	<ul style="list-style-type: none"> ▼ A faces is displayed in a color which corresponds to the operation resulted in this face. ▼ The operations, bodies, surfaces, curves and other model objects are displayed in a color specified for the model (or a variant, if any).
	By source available for operations and components.	<ul style="list-style-type: none"> ▼ Each operation is displayed in a color specified for a body (or a surface) built as a result of this operation. ▼ Each component is displayed exactly as it appears in its file.
	By layer available for objects that can be placed in layers. layers* (i.e. all objects except operations and faces).	The objects are displayed in colors specified for their layers.
	Manual setting available for all objects	Each object is displayed in a color specified by a user.

* For details about layers, see Section 2.15.13 on p. 845.

When setting colors and properties of surfaces, keep the following in mind:

- ▼ A color and properties of a model are always set manually. So, when you configure the group properties, the **Assign color** group is not available on the Properties panel, and only the **Color** field and the **Optical properties** panel are shown. When configuring all other objects, keep the following in mind:
 - ▼ The **Color** field is available only when the **Manually** switch is enabled.
 - ▼ The **Optical properties** panel is available, only when the **By source object**, **By layer** (in this case the properties cannot be edited and are given for information only) or **Manually** switch (in this case the properties are editable) is enabled.
- ▼ You can change the color and properties of the surface for the components of the first level only. When configuring the components of other levels, the controls for surface color and properties are not available on the Properties panel.



After the color surface properties have been adjusted, confirm the changes made.

After that, the image in the window will be re-drawn according to set parameters.

You can also change the color and properties of the object surface in the **Properties** window (see 2.14.3.6.1 on p. 793) by setting the appropriate values to the *Color designation*, *Color* and *Optical properties* properties.



In the **Properties** window, you can customize several objects at once. For example, you can select several edges of a part or components in the assembly and set their color.



Sometimes parallel edges of a body merge in halftone image. To facilitate perception of such an image, properties of parallel faces may have different settings (often it is sufficient to change shininess or diffusion without changing color).

2.14.3. Editing model objects

When editing a model, you can change any parameters of its nested objects. This section describes all editing methods available for the objects.

Note that you can edit only objects directly included into the current model. You cannot directly edit the objects of the components within this model. To edit the objects in the component, you can either open it in a separate window (see Section 2.11.6.1 on p. 722) or enable the context editing mode for the component (see Section 2.11.6.2 on p. 723). Then you will be able to edit objects without any limitations.

When editing objects in a model, the hierarchical relations between them are not changed, i.e. you don't need to reconfigure the sequence for the child objects and their parameters after changing the parent object.

In some cases, a model is rebuilt automatically. For example, when you change a contour in a sketch, the system automatically adjusts the shape of the underlying extrusion object. In other cases, e.g. when the values of variable are changed, you should call a certain command to rebuild a mode, see Section 2.14.3.7 on p. 795.

Changes that you make when editing a model are copied into all parent models.

2.14.3.1. Editing a Sketch

2.14.3.1.1. Editing an image in a model's sketch

You can edit a graphical image in any model sketch.

You need to select a sketch before calling the sketch editing command. This can be done in one of the following ways:

- ▼ Select the sketch in the Construction Tree,
- ▼ Select an operation based on the sketch in the Construction Tree,
- ▼ Select any face of an object based on the sketch in the Model Window

Then call the context menu command **Edit Sketch**.

The system enters the sketching mode.

At the same time, only those objects will be left in the part window that are located before the edited sketch in the Construction Tree. In other words, the model will temporarily return to the state in which it was at the time of the edited sketch creation.

When making changes to a sketch, you can project (map) on it the existing objects (edges, faces, etc.), snap to model objects, add and delete parametric relations and constraints, perform any constructions, edit graphical objects.

If the sketch is parametric and the object positions in it are defined by associative dimensions, you can enter new values for those dimensions.



Remember that the sketch for an operation must meet certain requirements. These requirements shall be met not only when creating the sketch, but also when editing it. Specific requirements to sketches are given in Sections devoted to operations.



After the desired changes to the sketch are made, leave the sketch editing mode. The object formed on the basis of the sketch and its derived objects will be rebuilt in accordance with the contour modification in the sketch.



We do not recommend making such changes to a sketch, that will certainly cause a failure in rebuilding dependent objects.

2.14.3.1.2. Positioning Sketch on the Plane

In some cases, it may be required to change the position of the sketch on the plane (flat face) or select another plane (flat face) for the sketch. To do so, select the sketch in the Design Tree and call the **Locate sketch** command. The sketch placement elements will become available on the Properties Bar (see Fig. 2.1.23 on p. 141).



If graphic objects in a sketch have parametric relations with an auxiliary model object projections (see Section 2.2.2.3.1 on p. 146), a prompt will be displayed about the possible loss of those relations after changing the sketch plane. You can continue positioning the sketch or cancel it.

To select another plane (flat face) to place the sketch, select it in the Model Window. The location of the sketch will change and the name of the selected object will appear in the **Base Plane** field on the Properties Bar.

For relocating a sketch on the plane, the **t** and **Angle** fields are used. You can specify the new position of the selected sketch's coordinate system and the rotation angle of the new coordinate system in relation to its current position. The new coordinate system's position can be adjusted in the window using the cursor. To do so, you first need to unfix the respective field — **t** or **Angle**. Upon unfixing either, the ghost of a coordinate system will appear on the screen.

The position of the sketched objects with respect to its own coordinate system will not change. Therefore, as the coordinate system moves, the sketch image would move as well. The changing position is displayed as a ghost.

If you do not need the sketch to be related to the base plane, enable the **Independent Location** option. The sketch will keep its position in space, but its link with the base plane will be broken (the base plane will no longer be highlighted in the Model Window, and its name will be deleted from the **Base Plane** field). When the option is enabled, sketch location parameters are unavailable.

Disabling the **Independent Location** option restores the link between the sketch and the base plane.



To perform all the required actions, click **Create Object**. The object formed on the basis of the sketch and its derived objects will be rebuilt in accordance with the sketch's new position.



The **Locate Sketch** command is unavailable if graphic objects in the selected sketch have a parametric relation with previously created objects (e.g. handles of sketched line segments match with vertices of the face on which this sketch is built, or the sketch contains projections of existing edges) or if other parametric relation types prevent from moving graphical objects in the sketch plane.

2.14.3.2. Operation editing

A model's object is created by performing an operation. You can edit the created object by modifying the operation's parameters.

In order to modify parameters of an operation, enter the object editing mode. For this purpose, do any of the following:

- ▼ Select an object in the Construction Tree or in the Model Window and call the **Edit** command from the context menu.
- ▼ Select an object in the Construction Tree or in the Model Window and call the **Edit** command from the **Editor** menu.
- ▼ Select any face, edge, or vertex of an object in the Model Window and call the **Edit source element** command from the context menu.

The system will switch to the command that was used for building the selected object.

At the same time, only those objects will be left in the part window that are located before the edited object in the Construction Tree. In other words, the model will temporarily return to the state in which it was at the time of the edited object creation.

The Property bar will display the same fields and option buttons for setting parameters of the operation as when constructing an object. If reference or source objects were used for object construction, they will be highlighted in the Model Window and in the Construction Tree.

Edit the required operation's parameters (create new numeric values of parameters, modify the set of reference or source objects and so on).



If **Sketch Dimensions in Operations** option is on in the dimensions and symbols display settings dialog (see Section 9.1.11.12 on p. 1736), then sketch dimensions will be displayed and available for modification when editing features. To adjust the dimension, double-click the dimension text.



Confirm the adjustments by clicking **Create Object** button.

The object will be rebuilt according to the new parameters of the operation.

2.14.3.3. Editing numeric parameters of an operation

You can edit number parameters of an operation without entering its editing mode. For this purpose, you can use variables, corresponding to parameters of the operation, and operation's dimensions (see Section 2.9.2.2 on p. 632).

More information about editing variables related to operation's parameters can be found in Section 2.14.3.3.1 on p. 789. Details of editing operation's dimensions can be found in Section 2.14.3.3.2 on p. 789.

2.14.3.3.1. Editing a variable's value

One of the methods to edit an operation's numeric parameter is changing its variable's value in the Variables window.

In order to change the value of a variable, do the following:



1. Enable displaying of the Window of Variables. For this purpose, click the **Variables** button on the Standard bar.
2. Locate and expand the section corresponding to the desired object in the table in the Variables window.
3. Find the line of the variable being edited.
4. Click the **Expression** cell of the line and enter new value of the variable.
5. Press <Enter>.



In order to rebuild the model in accordance with the parameter's new value, click the **Rebuild** button on the **View** bar or choose its name from the **View** menu.

2.14.3.3.2. Editing dimensions of operation

One of the methods to edit an operation's numeric parameter is changing the value of the respective dimension of operation.

You can edit operation's dimension in the special mode (see Section 2.9.2.2.2 on p. 635).

In order to modify the value of operation's dimension, do the following:



1. Enable displaying dimensions of the selected element. For this purpose, click the **Dimensions of selected element** button on the **View** bar or select its name in the **View** menu.
2. Highlight the required object in the Construction Tree or specify any primitive of this object in the Model Window: a face, edge, or vertex.
Dimensions of the highlighted object will be displayed.
3. Find the required dimension and double-click its dimension text. The dialog to set the dimension value will be displayed.
4. Enter the dimension's value in the **Expression** field of the appearing dialog: a number, constant, or expression for calculation of the dimension's value. Syntax of expressions is described in detail in Section 7.1.3.3.1 on p. 1580.



5. Click the **Rebuild** button in order to enable automatic rebuilding of the model after closing the dialog.
6. Finish dimension editing by clicking the **Create Object** button.
The dialog will be closed, and the model will be rebuilt according to the value of the parameter.

More information about the dimension value setting dialog and its usage can be found in Section 2.9.2.2.1 on p. 634.



The value of a sketch's dimension can be changed outside the sketch mode. Modifying the value of a sketch's dimension is the same as modifying the value of an operation's dimension.

If operation's dimension is represented by a control element (i.e. derivative dimension is created see Section 2.9.2.8 on p. 648), you can edit the operation's parameter by changing the respective derivative dimension's value. Editing the value of the derivative dimension is the same as the editing procedure described above.

2.14.3.4. Editing Models Inserted from Library

After a library model is inserted in an assembly, you may need to edit it.

- ▼ If, on the other hand, the selected model does not have external variables, then to modify it you would need to edit the source model in the library (see Section 12.1.5 on p. 2045).

Refreshing model inserts (whether from libraries or models from disk) occurs automatically upon opening the main document.

- ▼ If a model inserted from the library has external variables, then, you need to edit their values in order to modify dimensions and the model topology (see Section 7.1.5.3.2 on p. 1593).

If a library model has a table of variables in addition to external variables, you can select new set of variable values from the table. You open the table of variables using **Table of Variables** command in the context menu of the library model in the Model Tree.

Sometimes, after editing the variables of a model, the latter is marked in the Design Tree as failing. That means the input external variable values make the system of equations relating the model variables unsolvable. To fix the error, edit variable values again.

- ▼ You can also replace a library model with a model from the disk (select another model insertion source).

To change the model insertion source, select it in the Design Tree and call the context menu command **Component Properties**. The dialog of new library settings will be displayed. The **Source File** field shows the path to the library file, from which the model was taken, and the model's name in the library. In order to change the source of the model's insertion, click the **Change...** button. In the appearing standard File Open dialog, select the desired model document.



A library part component can be replaced by a part only, an assembly component — by an assembly only.

The selected model will be inserted in the assembly instead of the library component, while maintaining the anchor point position. The library component icon will disappear in the Design Tree, instead, a part or assembly icon will appear in that place.



If an assembly has objects derived from a library model (for example, a construction axis was drawn through one of its vertices, or a part was built in the assembly context on one of the library model's faces), then, upon replacing the model insertion source, all relations, which arose from creating dependent objects, will be destroyed, and errors will appear in the model. Therefore, before you change the model insertion source, review this model relations to determine the derivative elements it has.

2.14.3.5. Features of Editing Individual Objects

The techniques of editing most features (deletion, parameter setting, selecting another reference objects, etc.) are the same. They are described in the previous sections.

However, certain objects have specific properties meaning there are additional techniques for editing such objects. They are described in this section.

2.14.3.5.1. Hole from a library; spiral; hole in sheet body

One of the features of the objects like a hole from a library, spiral and hole in sheet body is that when you create them, the system automatically generates a sketch in the model. The sketch contains an auxiliary point at the intersection of the object's axis and an edge (or plane) where it resides. We will call this point a central point of an object.

If the sequence of model building is displayed in the Tree (see Section 2.1.2.1 on p. 106), the position of objects in it will correspond to the order of creation, and the sketch with the center point will be positioned directly in front of the hole (or a spiral).

If the Tree displays the model structure (see Section 2.1.2.2 on p. 106), the sketch will be positioned in the Sketches section, whereas the hole will be located in the Body Section (or in Curves and Points section for spirals). In this case, the relations of the *Hole* object or *Spiral* (see 2.1.2.7.2 on p. 114) can be used for quick search of the sketch with the hole central point. The required sketch will be the first in the list of source objects *Holes* or *Spirals*.

Using this sketch, you can do the following:

- ▼ Change the position of an object on a face. To do this, select a sketch with a central point in the Design Tree (or the point itself in the model window) and then choose the **Edit** command from the context menu. The system will switch to the sketch editing mode. Move a point from the sketch to the destination using any method you like. You can use bindings, apply parametric limitation to the point or set the parametric dimensions to define the position of the point.

For example, if you need the hole to be positioned exactly in the center of a rectangular face, create an auxiliary line connecting the diagonal points of the face and snap the point to its middle. After that, the center of the hole will stay exactly at the center point of the face throughout any changes in the face dimensions.



After the desired changes to the sketch are made, leave the sketch editing mode. The object will be rebuilt to accommodate the new center position.

- ▼ Move an object to another face or surface. To do this, select a sketch containing the center point from the Design Tree and call the **Place sketch** command (see Section 2.2.2.2 on p. 145).

2.14.3.5.2. Dimensions and Symbols

In contrast to other objects in the model, dimensions and symbols can be edited without invoking the edit command, just using the mouse.

Select the dimension or symbol by clicking it in the model window or Construction Tree. The model window will show the selected object's handles. You can move them with your mouse in the dimension plane or symbol plane. In such a way, for example, you can move a dimension line or text, rotate the leader shelf, change the spot where the leader branching is pointing at, etc.

If a symbol is selected, this symbol's plane will also be displayed as a dashed rectangle. If a symbol plane was not fixed when the symbol was created (i.e. no fix object was specified), then one of the corners of the rectangle will show a handle that allows moving the symbol plane parallel to itself. When this handle is dragged with the mouse, the whole symbol will move. The plane can be moved as long as the condition is met that all starting points of all branchings belong to the object which the symbol was attached to.



If the symbol plane cannot be moved (e.g. the symbol is attached to a point object), the symbol will remain in its place.

If you need to edit text as part of the object (dimension text or text in designation), select the object, move the cursor away from this object and press `<Shift> +<Enter>`. Make changes in the appearing dialog and then close it by clicking the **OK** button.

You can call the dialog for setting the dimension value by double-clicking the dimension text. Depending on the dimension's type, you can use this dialog to change name of the variable, dimension's value, and other parameters (see Section 2.9.2.2.1 on p. 634).

2.14.3.5.3. Editing Mates

To edit a mate, select it in the Design Tree and use context menu item **Edit**.



Unlike the editing of operations, when editing a mating, the model is not refreshed to the state it had at the time this mating was created.

If necessary, specify other objects for mating. Adjust the mating parameters (such as object orientation or distance between them) using controls on the Property bar.

If the shape of the selected elements and the existing relations and constraints do not conflict with the mating, then the model will attribute in such a way as to satisfy the mating condition. If the mating cannot be rendered with the specified conditions, then the system will mark it as incorrect just after the editing. Sometimes the mating condition may be rendered correctly however making conditions of other matings incorrect. In this case the error will be displayed in these matings.



When editing a mate, you cannot change its type, that is, you cannot turn, for example, an *At Distance* mate into a *Parallel* one.

2.14.3.6. Other techniques of editing

2.14.3.6.1. Editing a model using the Properties window

You can view and change the following properties of selected objects without editing each of them individually:

- ▼ display parameters – color, line style, vertex style, etc.,
- ▼ properties – symbol, name,
- ▼ state — managing fixing, visibility, calculation, editing, projecting objects into the associative drawing,
- ▼ parameters set during object creation (for dimensions, symbols, and so on),
- ▼ the text included into the object contents,
- ▼ the text style, font and paragraph parameters,
- ▼ spelling language attribute.



In order to change object properties, as well as to view the properties window, use the **Properties** window. This window is displayed by calling the **Editor – Properties** command or by clicking the **Properties** button on the Standard bar.

Controls of the **Properties** window and modifying the properties are described in Section 1.4.2.2 on p. 82.

When working with properties in the model documents, keep the following in mind.

- ▼ You can specify objects of the first level – axis and coordinate systems, layout elements, curves and points, sketches, objects of auxiliary geometry, components of the first level of bodies created in the model, faces of the bodies, etc.
- ▼ The list of properties in the window depends on the type of the selected object. For example, clicking the dimension's extension line displays its attributes as a layout element. Clicking the dimension text displays properties of the dimension text.
- ▼ You can set the object's property to **By layer**. In this case, the value of the property inherits the value set for the layer where the object resides. The layer-controlled properties of the objects include:
 - ▼ visibility,
 - ▼ ability to edit,
 - ▼ color,
 - ▼ ability to project into the associative drawing.
- ▼ Values of properties in the **Properties** window are available for editing if this does not conflict other properties. For example, you cannot change the **Color** and **Optical properties parameters**, when the color is set **By source object**, **By source** or **By layer**.
- ▼ The **Visibility** property of an object is not available when all objects of this type are hidden using the **Hide all auxiliary objects** command (see Section 2.14.1.1.2 on p. 781).
- ▼ Changing the **Base designation** property is not available, if automatic sorting of letters in the designation was enabled during creation or editing the base (see Section 2.9.3.3 on p. 658).
- ▼ The **Language** property is available, if designation of a leader is highlighted.

2.14.3.6.2. Editing model

Using End of Model Design Marker

By using the End of Model Design Marker (see Section 2.1.2.6 on p. 112), you can suspend objects from the model. The Marker can also be used for changing the model design sequence.



You will be able to carry out editing using the Marker if Model Tree shows the model design sequence (see Section 2.1.2.1 on p. 106).

Move the Marker so to make it split the Construction Tree into two portions.

You will see that the objects that are below the Marker are displayed in the grey color and are marked with a “lock” icon. These are suspended objects. Information about them remains in the document, however, these objects and their derivatives are not displayed in the model window.

To quickly move the Marker to the end of the Construction Tree, you can use **Rollback Bar to End** command from the Marker’s context menu. This command restores all suspended objects in the model. The model will be rebuild taking into account the restored objects.

Please note the following features of suspended deletion and restoration of objects in the model.

- ▼ Object restoration (by moving the Marker to bottom) is actually re-creation of these objects. In particular, this has the following consequences.
 - ▼ If any changes were made to the properties of faces of a feature prior to its suspended deletion, then these changes will be lost after the feature is restored, because the feature will be re-created with default face properties.
 - ▼ If the model had objects derived from instances of a component array (a chamfer was constructed on the instance edge, a sketch was constructed on the face, an auxiliary axis was based on the vertex, etc.) when the component array was suspended, then these objects will be marked as failing objects after restoring the array. The cause will be that the previous array instances (which the objects were based on) are no longer in the model: they have been replaced by new ones. The failing objects should either be deleted and then re-created, or edited by specifying new source objects for them.
- ▼ If the Marker is moved above the component array while working with the model, a delete objects dialog will appear listing all components-array instances to be deleted and the matings (if any) which they are involved in.
 - ▼ To finalize moving the Marker, confirm the deletion by clicking **OK** in the dialog. Please note that mating information will be deleted from the model, i.e. you will not be able to recover the mates just by moving the Marker. Array instances will be suspended and can be restored.
 - ▼ To cancel moving the Marker, decline the deletion by clicking **Cancel** in the dialog.

While working with a model, you are not in any way limited to move the End of construction marker. The icons of the newly created elements will be positioned in the Construction Tree before the Rollback Bar.

This property of the End of Model Design (rollback) marker, and that the excluded from calculations elements are not displayed on the screen and cannot be used in constructions is convenient to use when modifying the order of building the model.

For example, to edit a part obtained by the operation of mirror copying, you need to make changes to that its half, which is the original, by positioning them before the mirror copy operation. However, firstly, both halves of the part will look completely identical. Secondly, while editing, you can accidentally use the objects of the part's half that was created by copying, which will inevitably lead to an error. In such cases, it could be helpful to move the End of Model Design (rollback) marker so that the mirror copy operation and all the following ones were excluded from calculations. Only the original half of the part will be displayed. You can edit it as desired. After including the copy operation into the calculations, the second half of the part will also include the introduced changes.

If the model elements located below the End of Model Design (rollback) marker are not needed for further construction and do not have to be in the model, you can simultaneously delete all those objects. To do this, use the command **Delete Elements below Rollback Bar** (the Marker) from the context menu accessed on the Marker.

Upon calling the command, a warning dialog will appear. It will list model elements to be deleted. You can cancel or confirm the deletion.

2.14.3.6.3. Changing the sequence of operations

A convenient (though rarely used) technique of editing a model is “dragging” operations in the Design Tree. In this way, you can quickly change the design sequence.



The design sequence can be modified if the model design sequence is displayed in the Model Tree (see Section 2.1.2.1 on p. 106).

To move an operation in the Design Tree, hover the cursor over its icon, left-click it and, while holding it, drag the cursor to the desired location in the Design Tree. While doing so, the cursor appears as an arrow indicating the operation's position.



When the desired operation location is reached, release the mouse button. The operation's icon is placed in the Design Tree at the specified position and is marked by a red “check mark”. This means the change in the creation order has not been yet propagated to the model — the image in the window remains the same.



To rebuild the model, apply the following command: **View— Rebuild**.

The operations are moved with respect to their hierarchy. An operation cannot be moved about the source (parent) one or below a dependent object in the Design Tree.

2.14.3.7. Model rebuilding

Rebuilding a model can be considered as making the model up-to-date. When you rebuild a model, the shape and position of its objects are recalculated, so that the model appearance on the screen can change.



If MP parameters are recalculated during rebuilding (see Section 9.1.11.15 on p. 1738), then the mass value and coordinates of the center of gravity are also updated within the model.

Rebuilding is required when the model contains conflicts, i.e. its geometry does not match the original data, such as history of builds, operation parameter values, matings, etc. The conflicting objects are marked with a red “check mark” in the Model Tree. When at least one object requires rebuilding, the model's header in the Tree is also marked with the red “check mark”, so that you will see it even when the Tree is fully collapsed.



To rebuild the model, choose **View — Rebuild** or press <F5>. The button to call this command is located on the panel **View**.

The conflict in the model can occur when you make changes in one of the following objects:

- ▼ The value of a variable corresponding to the operation parameter was changed in the Variables window. The resulting object will not change in the model window, but will be marked with the red “check mark” in the Design Tree. After rebuilding the object in the model window will be updated with the new value of the parameter and the red “check mark” will be removed.
- ▼ You've changed the sequence of the operations by dragging them with the mouse in the Design Tree. All operations you've moved are marked with the red “check mark” in the Tree, but the model in the window does not change. After rebuilding the “check marks” are removed and the appearance of the model may either change or stay as it is depending on the set of operations. For example, if you drag a trimming operation in between the plains outside these plains, then the body will be trimmed entirely. However if you swap two independent operations in the Tree, the model will likely stay unchanged.
- ▼ Two components are mated in such a way that a face of the first component touches a cylinder surface of another one. Then you've edited the second component, so that its cylinder face used in this mating turned into a conic one. The new shape of the component will appear in the model window, but the position of the first components will not change. The icon of the edited component is marked with the red “check mark” in the Design Tree. After rebuilding the “check mark” disappears and the first component is positioned in such a way that the existing mating (touching of the surfaces) remains consistent.
- ▼ The model contains an auxiliary axis going through the vertices of the two components. Then the position of one of the components was changed, e.g. due to mating with another one. The icon of the component is marked with the red “check mark” in the Design Tree, and the auxiliary axis does not change its position. After rebuilding the red “check mark” disappears, and the position of the axis is changed in such a way that it goes through the specified vertices.

In some cases, the system can automatically determine that the model needs rebuilding and notify you about it. For example, this request may occur when you load a model in which one of the components does not match the data stored in this model. This may be caused either by changes in the component's geometry or changes in its properties, such as name, designation, material, etc.

The conflicting components are displayed in the request window.

If you want to rebuild the model, click the **Yes** button of the dialog.

If a model consists of a large number of components, rebuilding it can take a significant time. Therefore, if you need to edit several components of a complex model, it would be rational to

rebuild it just once — after making all changes (rather than each time after editing an individual component).

To postpone rebuilding the model, click the **No** button of the dialog. The model rebuild request will disappear, and the icons of the objects to be rebuilt will be marked with the red “check mark” in the Design Tree.



You can also rebuild the model in any time by choosing **View — Rebuild**.

After rebuild some models may display errors, such as exclamation marks in red circles, which mean that the objects are built incorrectly. For example, suppose, a model cut element was extruded up to a face of some part. Then this part was displaced so that the mentioned face could not further constrain the extrusion element (i.e., the element sketch is either partially projected on this face or it cannot be projected on it at all). The element cutting become impossible, and after the model rebuilding this operation is marked in the Design Tree as wrong.

2.14.3.8. Deleting an object

Any object (a geometry-producing feature, a sketch, an auxiliary axis or plane, assembly component, a mating, etc.) can be deleted from the model. You can do this simply by selecting them in the Construction Tree and using context menu item **Delete** or pressing <Delete> key.

If other objects reference the deleted object or the deleted object is involved in mating, then a warning dialog will be displayed. It displays the hierarchical list (see Fig. 2.14.4) where:

- ▼ the first level – icon of a model or variant whose objects are being deleted;
- ▼ the second level – **objects that are being deleted**;
- ▼ the third and the following levels – model's objects related to objects being deleted (linked objects also include mates with participation of objects being deleted); linked objects are **deleted together with their source objects**.

You can cancel or confirm the deletion.

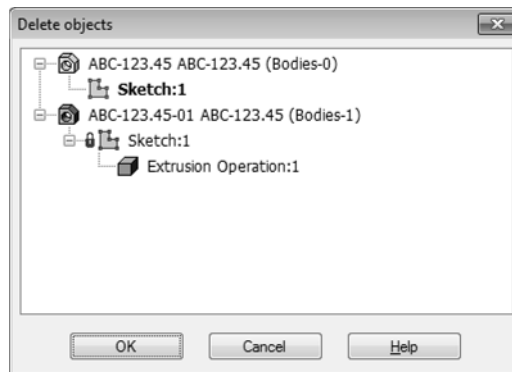


Fig. 2.14.4. Items Deletion dialog

2.14.4. Restructuring a model

Editing model structure means change of the components hierarchy. In most cases, a set of files comprising the model is changed as a result of assembly structure editing.

The following options of model structure editing are available:

- ▼ part– subassembly transformation and subassembly– part transformation (see Section 2.14.4.1),
- ▼ integration of components into subassembly and explosion of subassembly (see Sections 2.14.4.2 on p. 799 and 2.14.4.3 on p. 799),
- ▼ copying and moving components (see Section 2.14.4.4 on p. 800),
- ▼ splitting model objects into parts (see Section 2.11.2.3 on p. 689).



As a result of structure editing, certain matings laid upon the components participating in this procedure can be removed from the model. Therefore, it is recommended to set matings after complete specification of the model structure.

2.14.4.1. Part-to-subassembly and subassembly-to-part transformations

- ▼ Part-to-subassembly transformation means saving the part inserted into the current model in the file of a new assembly (*.a3d).
- ▼ Subassembly-to-part transformation is saving of the subassembly inserted into the current model in the file of a new part (*.m3d).

You may need part – subassembly and subassembly –part transformations when designing assembly 'top down'. For example, at the initial stages of a gearbox design, it was planned that its cover would be welded. Correspondingly, the *Cover* was inserted into the gearbox model as a subassembly. It turned out later that it would be wiser to manufacture a cast cover, i.e., the component should be a part and not a subassembly. To quickly edit the gearbox model, use subassembly-part transformation.

The transformation is only applicable for the components at the first level of the model structure, i.e., for those inserted directly into the current model. When you need to transform the components built into a subassembly, you should enter the editing mode of this subassembly.

To perform the transformation, select the required component (a part or an assembly) in the Design Tree and call the **Save As...** command from the context menu or go to **Editor — Edit component**.

Windows-standard File Save dialog will be displayed. In the **File Type** list select the following:

- ▼ to convert a part into a sub-assembly— the **KOMPAS-assemblies (*.a3d)** line,
- ▼ to convert a sub-assembly into a part — the **KOMPAS-parts (*.m3d)** line,

Specify both folder and file name for saving, then click the **Save** button. The dialog will close.

The saved component replaces the previous one in the current model. The previous component will be removed from the model, but its file on the disk will not be changed.



To save the transformations in the current model, choose **File— Save** command.

The subassembly – part transformation has the following peculiarities.

- ▼ All objects created in the subassembly are saved in the part file: bodies, sketches, auxiliary objects, etc. Also, the following objects created in the components of a subassembly are copied to the part:

- ▼ bodies, surfaces, curves – as the respective objects without history,
- ▼ points, including control and connection ones – as points constructed based on coordinates.
- ▼ The following errors may occur when transforming a subassembly into a part:
 - ▼ errors related to missing reference objects, when the components of the transformed subassembly were used for building other objects in the model (including those located in the same subassembly),
 - ▼ mating errors – when components of the transformed subassembly were used in matings.



The **Save as...** command can be used to save the component under a different name and to automatically insert the renamed component instead of the previous one into the current model.

2.14.4.2. Combining components into subassembly

You can combine several components of the current model into a subassembly.

It is possible to combine only components of the first level, i.e. inserted directly into the current model, not in any of its sub-assemblies.

The product of the combination is a new assembly file (*.a3d) containing the components of the current model selected for combining. The new assembly will be inserted in the current model as a subassembly, and components it will be made of will be deleted from the model. Files of these components remain unchanged on the disk.



The absolute coordinate system of the new subassembly will match the absolute coordinate system of the current model.

To combine components into a subassembly, select them in the Design Tree and call the context menu command **Combine into subassembly**. In the displayed standard file save dialog box specify both folder and assembly file name for saving, then click the **Save** button. The dialog will close.



To save the transformations in the current model, choose **File— Save** command.



It is impossible to combine into a sub-assembly the components being instances or source objects of an array.

2.14.4.3. Exploding subassembly

You can explode a subassembly of the current model into individual components.

It is possible to explode only the first level subassemblies, i.e. inserted directly into the current model, not in any of its subassemblies.

As a result of subassembly explosion, all its components are transferred into the current model. Their position is not changed. The other objects of the subassembly— surfaces, curves etc., as well as operations made in the sub-assembly (bevels, openings etc.) — are not transferred to the current model.

Destroyed sub-assembly is deleted from the model. File of sub-assembly on the disk will not change.

To explode a subassembly, select it in the Design Tree and call the **Explode component** command from the context menu or from the menu **Editor — Edit component**.



Sub-assemblies being inside the destroyed sub-assembly are not destroyed. If necessary, you can apply the explode command to them.



It is impossible to explode the sub-assembly being an source array object.

2.14.4.4. Moving and copying components

You can move and copy components:

- ▼ from one to another subassembly of the current model,
- ▼ from the current model to any of its subassemblies, and visa versa.

Moving is performed in the Design Tree by “dragging” a component with the mouse to the sub-assembly or model icon (or to the icon of the *Components* section inside the subassembly or model).

Copying of a component is performed in a similar way, but with <Ctrl> button pressed; in doing so, the “+” sign is displayed next to the cursor.



When “dragging” a component over the icon of an object to which it cannot be moved or copied, a warning sign is displayed instead of the cursor.

After the component has been moved, its position in the model is not changed.

As a result of copying, overlaying components are obtained. To change the position of the source component or the copy component, use the move and rotate commands (see Sections 2.11.3.1 on p. 699 and 2.11.3.2 on p. 699).

You can copy and move components not only in the Design Tree but in the Model window as well. In this case, you can randomly set the position of the component in the model's coordinate system. However, the position of the component in the model structure is limited to the first level, i.e., the moved or copied component is inserted directly into the current model.

To move a component in the Model window, do the following.

1. Select the component in the Design Tree or in the Model window.
2. In the model window, place the cursor on the component.
3. Click the left mouse button and press the <Alt> key.
4. Move the mouse. The ghost of the component will move together with the cursor in the Model window.
5. When the ghost reaches the desired position, release the mouse button and then the <Alt> key.

Copying of a component is performed in a similar way, but with the <Ctrl> button pressed; in doing so, the «+» sign is displayed next to the cursor. In addition, for copying you can select not only one but multiple components as well.

You can also copy components using their icons in the Design Tree. For this, do the following.

1. Select the component(s) in the Design Tree.
2. press the <Ctrl> key and “drag” the component from the Design Tree into the Model window. Release the mouse button.
3. Move the mouse in the Model window. The ghost of the component will move along with the cursor.
4. As soon as the ghost reaches the required position, click the left mouse button and release the <Ctrl> key.

No matter what method is used to move or copy the component, this operation will lead either to creation or removal of a reference to the component file in the model file:

- ▼ when the component is moved, a reference to it is removed from one file (for example, subassembly file) and added to another file (for example, assembly file),
- ▼ when the component is copied, a reference to it is added into model or subassembly file.



You cannot move the component that is a source array object or instance.

2.14.5. Saving an assembly as a part and a part as an assembly

You can save the current part (*.m3d) in the new assembly file (*.a3d), i.e. transform the part into an assembly and vice versa, as well as save the current assembly (*.a3d) in the new part file (*.m3d), i.e. transform an assembly into a part.

To perform the transformation, select **File — Save as...**

Windows-standard File Save dialog will be displayed. In the **File Type** list select the following:

- ▼ to transform part into assembly— line **KOMPAS-assemblies (*.a3d)**,
- ▼ to transform assembly into part— line **KOMPAS-parts(*.m3d)**,

Specify both folder and file name for saving, then click the **Save** button.

Upon the completion of the transformation, the source part or assembly is still displayed on the screen. To download the model which is the product of transformation, use the **File-Open** command.

When transforming an assembly into a part, keep the following in mind.

- ▼ All objects created in the assembly are saved in the part file: bodies, sketches, auxiliary objects, etc. Also, the following objects created in the components of an assembly are copied to the part:
 - ▼ bodies, surfaces, curves — without history,
 - ▼ points, including the reference points and connection points — with coordinates.
- ▼ A resulting part may contain errors due to missing reference objects, if its components (such as axis going through the zero points of the components) were used for building the assembly's components.



If you want to convert a child model in order to replace it, choose **Save as....** (see Section 2.14.4.1 on p. 798).

2.14.6. MP and model's hatch parameters

The mass properties of models may be calculated or set manually.

- ▼ In the first case, the model's MCC are calculated based on MCC of its elements: bodies and components.
- ▼ In the second case only the center of gravity coordinates are calculated (the model mass is entered by the user) or no calculations are performed (both the model mass and center of gravity coordinates are entered by the user).

The model mass and center of gravity coordinates (either calculated or entered manually) are saved in its file. During the insertion of a model into another model as a component, these parameters are often sent to a model and used for calculating its mass properties.

If necessary, you may set the MPs for components which differ from their original ones. These parameters will be stored and used in the model which contains components.



To speed up MP calculations for a model, it is recommended to save all its components files in KOMPAS-3D version 10 or later.

The general methods of setting the MP parameters are covered in Section 2.14.6.1 whereas the procedure for setting MPs of bodies, components and models is given in Sections 2.14.6.2–2.14.6.4. Section 2.14.6.5 is dedicated to settings of a system and document related to the models' MP; section 2.14.6.6 describes setting model's hatch parameters.

2.14.6.1. General Methods of Setting MP Parameters

The mass properties of an object (body, component or model) are managed using the controls on **MP parameters** of the Properties bar.

The body volume is always calculated by the system automatically. The controls on **MP Parameters** tab may be used to set:

- ▼ material and density of material (see Section 2.14.6.1.1) to calculate the mass the center of gravity coordinates,
- ▼ arbitrary values of mass and center of gravity coordinates (see Section 2.14.6.1.2).

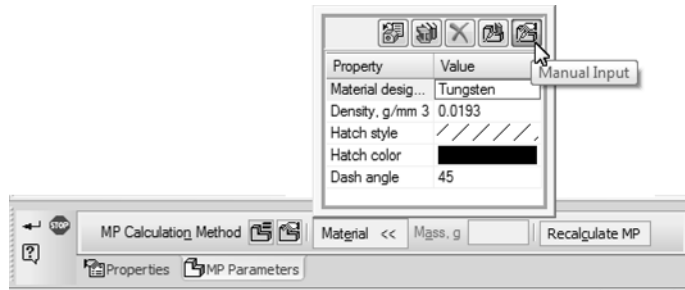
The **MP parameters** tab also contains **Recalculate MP** button (Fig. 2.14.5). Clicking this button initiates calculation of assembly MPs using the current values of parameters. The Message window with the calculation results pops up when calculation is finished



The Informational window contains the summary of calculation (mass, volume, area, center of mass coordinates of the body or the model). To obtain the inertia properties of the model (moments of inertia and direction of inertia main axes), use **Model Mass Properties** command (see Section 2.15.2.4 on p. 817).

2.14.6.1.1. Material and Material Density

Materials are selected using **Material** panel (Fig. 2.14.5).

Fig. 2.14.5. MP calculation by density; **Material** panel

The table displays properties of a model's material: name of the current material, density and hatch parameters that are used for displaying the material on cut/section in the drawing.



Select from List of Materials button allows selecting material from the density reference file (file ...IASCONKOMPAS-3D V...Sys\Graphic.dns; the values of the material's parameters contained therein may be edited by the user) whereas **Select from Reference of Materials** button enables selection from the Reference of Materials and Assortments.

Once the material is selected, values in **Material designation** and **Density** cells are filled in automatically.



To set density click the **Calculation by Density** radio button in the **MCC Calculation Mode** group on the Property Bar (Fig. 2.14.5). You can use the reference density value for calculation, i.e. the value which corresponds to the selected material or an arbitrary value.

▼ If the **Select from reference** switch is enabled, the reference density value is used in the calculation (the default option). The material density value will be imported from the density reference file or the Reference of Materials and Assortments and displayed in the **Density** field.

▼ To set an arbitrary density value, click **Manual Entry** radio button and enter the required value in the **Density** field.

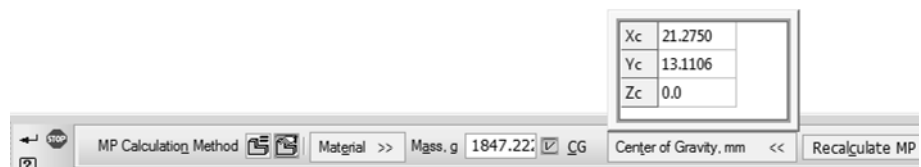
Refer to Section 2.14.6.6 on p. 809 for more information about models' hatch settings.

2.14.6.1.2. Mass and Center of Gravity Coordinates



To set the mass of a model, enable the **Calculation by Mass** switch (Fig. 2.14.6). Enter the mass value into the **Mass** field.

If necessary, you may also define the coordinates for the center of gravity. For that, enable the **CG** option and enter the coordinates into the table on the **Center of Gravity** bar.

Fig. 2.14.6. MP calculation by density, **Center of gravity** panel

2.14.6.2. Body MP Parameters Calculation

To setup the body MCC calculation parameters select it in the Construction Tree and call **Body Properties** command from the context menu (Fig. 2.14.7).

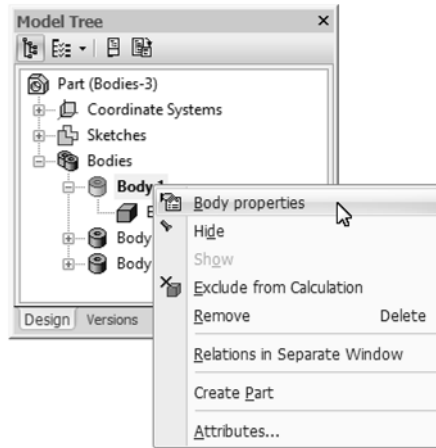


Fig. 2.14.7. Calling the **Body Properties** command



The body icons and names are shown on the Tree if the model structure is enabled (see Section 2.1.2.2 on p. 106).

Enable **MP Parameters** tab on the Properties panel (Fig.2.14.8).

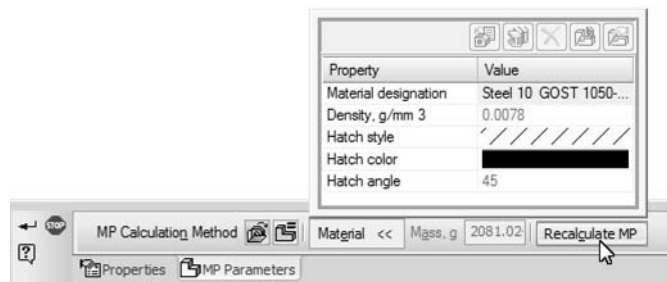


Fig. 2.14.8. Setting MP Parameters for a Body

The following MP parameter options are available for bodies.

- ▼ Using the MP parameters defined in the properties of the source model – i.e. the part of assembly on which the body being adjusted is built. To enable using the MP parameters of the source click **From Source** radio button in **MP Calculation Method** group. After this, **Material** and **Mass** controls will become unavailable.
 - ▼ Setting Body Material and Material Density (see Section 2.14.6.1.1).
- After completing MP Parameters setting click **Create Object** button on the special control panel.

2.14.6.3. Parameters for calculating components MP

The MP parameters may be set for any element in a model: a part (including local ones), sub-assembly, or a library component. While doing so the component file remains unchanged. You may also set up the MP parameters for components which constitute the array instances.

To set component MP parameters select the component in the Construction Tree and call **Component Properties** command from the context menu. Activate **MP Parameters** tab on the Properties panel.

The following MP parameter options are available for components:



- ▼ Using the MP parameters stored in the component source file. To enable using the source MP parameters select **From Source** radio button in **MP Calculation Method** tab. After this, **Material** and **Mass** controls will become unavailable.
- ▼ Entering the component mass and, if necessary, the center of gravity coordinates (see Section 2.14.6.1.2 on p. 803) (Fig.2.14.9). Please note that the component center of gravity coordinates must be set in the coordinate system.

If the component's mass center coordinates are not set by the user, they are calculated based on densities and volumes of bodies and components included in the component.

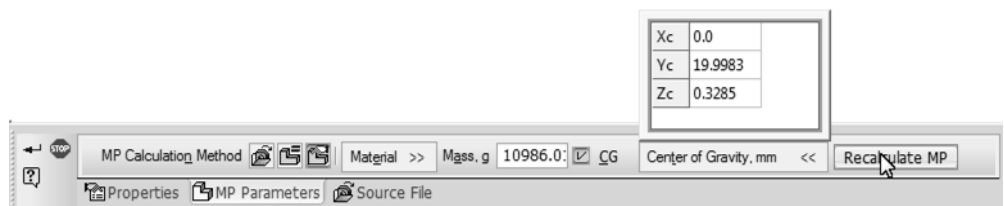


Fig. 2.14.9. Setting component mass and center of gravity

- ▼ Setting material and density for the bodies in a component (see Section 2.14.6.1.1) (Fig. 2.14.10).). The specified density will be applied to all the part bodies which do not have their unique density settings.

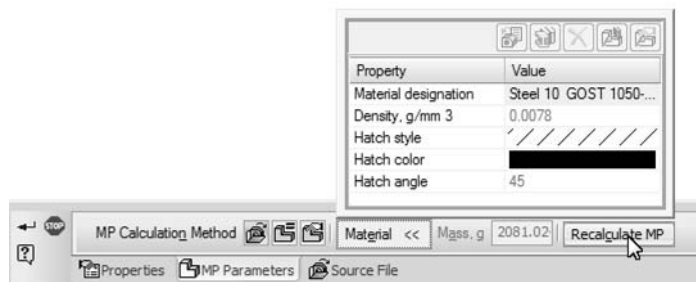


Fig. 2.14.10. Manual setting of component-part density



After finishing setting component parameters for MP calculation, click **Create Object** button.



In the Property Editor, you can view properties of materials of all bodies and components. You can also change their values for components of the first level and recalculate the mass of the model (see Section 5.1.4.2 on p. 1389).

Example of manual setting of component MP

Let us assume that a hypothetical, i.e., without drawing particular parts and elements, model of an electric motor is embedded into the assembly, and its mass and center of gravity coordinates are different from the standard ones. However, to obtain the correct MP of the assembly, it is necessary to use the standard MP parameters of the electric motor.

This can be done in two ways.

Method 1. Entering and storing the standard MP parameters in the electric motor model. For this, do the following.

1. Open the electric motor model, select the first element of the Design Tree and call **Model Properties** command from the context menu.
2. Go over to **MP Parameters** tab on the Property panel and click **Calculation by Mass** radio button in **MP Calculation Method** group.
3. Enter the standard mass value of the electric motor in the **Mass** field.
4. Activate **CG** option and enter the standard center of gravity coordinates of the electric motor into the table on **Centre of Gravity** panel.
5. Click **Create Object** button.
6. Save the model.

The specified MP parameters will be saved in the model and transferred to the assembly. Editing the model — adding or removing material — does not affect the mass value and center of gravity coordinates entered manually.

Method 2. Entering and storing the standard MP parameters in the electric motor assembly. For this, do the following.

1. Open the assembly containing the electric motor model as a component.
2. Select *Electric Motor* component in the Design Tree of the assembly and call **Component Properties** command from the context menu.
3. Go over to **MP Parameters** table on the Property panel and select **Calculation by Mass** radio button in **MP Calculation Method** group.
4. Enter the standard mass value of the electric motor in the **Mass** field.
5. Activate **CG** option and enter the standard center of gravity coordinates of the electric motor into the table on **Centre of Gravity** panel. The values of coordinates be specified in the component system of coordinates (not those of the assembly).
6. Click **Create Object** button.
7. Save the assembly.

The specified MP parameters of the component will be saved in the assembly and will be used for calculating its mass properties. There will be no changes in the electric motor model — the calculated mass values and centre of gravity coordinates will be stored unchanged.

Peculiarities of calculating model's MP parameters after editing

- ▼ The mass *set for a component manually* (in a model or in a source file) is considered fixed until the component or any of its parts is present in the model. If the model is edited so that the component's entire body disappears, it will be considered to have zero mass. For example, after creating a hole in a model going through a component, the mass of this component will not change; once the component is deleted by means of cutting by plane, the mass of the component will be zeroed.
- ▼ If a boolean operation is applied to bodies of two components in a model, then the mass values of these bodies will no longer be taken into account when calculating the model's MP parameters. Instead, mass of the body resulting from the boolean operation will be taken into account. If necessary, parameters of MCC calculation for this body can be changed (see Section 2.14.6.2 on p. 804).

2.14.6.4. Parameters for MP calculation of part and assembly

To set up the parameters for model (part or assembly) MP calculation select it in the Model Tree and call **Properties of a Model** command (Fig.2.14.11).

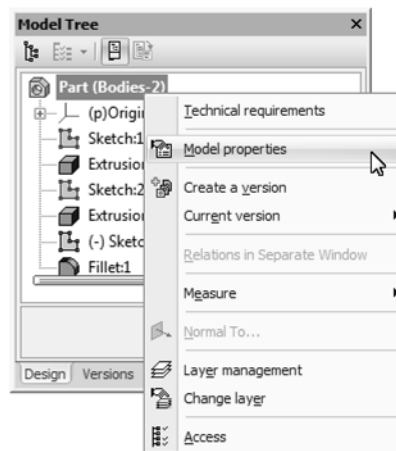


Fig. 2.14.11. Calling the **Properties of a Model** command

Activate **MP Parameters** tab on the Properties panel.

The following MP parameter settings are available for models:

- ▼ Setting material and density (see Section 2.14.6.1.1 on p. 802) (Fig.2.14.12). The specified density will be applied to all the model bodies created directly in the model and which do not have their unique density settings. The model mass is defined as the combined mass of its bodies and components.

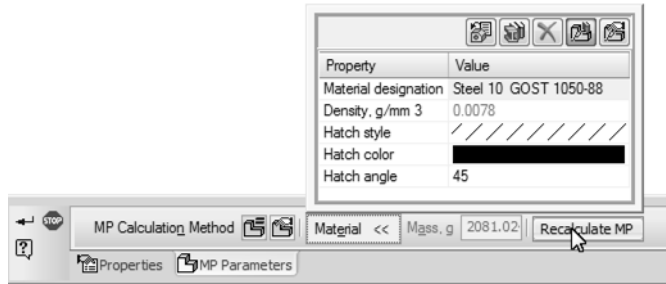


Fig. 2.14.12. Setting model density

- ▼ Setting the total model mass and, if necessary, the center of gravity coordinates (see Section 2.14.6.1.2 on p. 803) (Fig. 2.14.13). If the model's mass center coordinates are not set by the user, they are calculated based on parameters of bodies and components included in it. The bodies with no unique density settings are applied the reference value corresponding to the model material.

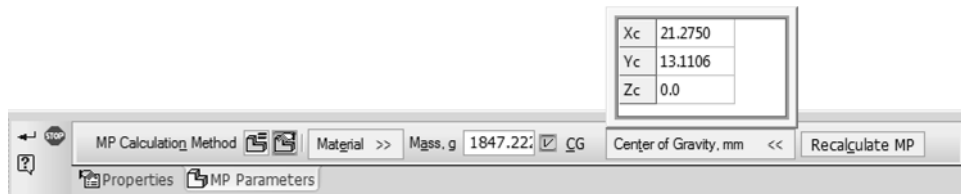


Fig. 2.14.13. Setting component mass and center of gravity



After completing MP Parameters setting click **Create Object** button on the special control panel.

The values of model mass and center of gravity coordinates calculated or entered manually will be recorded to the model file.



In the Property Editor, you can view properties of materials of all bodies and components in the model. You can also change their values (note that you can change the component's properties only for the components at the first level in the hierarchy) and recalculate the mass of the model (see Section 5.1.4.2 on p. 1389).

2.14.6.5. MP Setup

The following settings related to mass properties of models are available in KOMPAS-3D:

- ▼ Automatic MP calculation setup (see Section 9.1.11.15 on p. 1738),
- ▼ MP calculation precision setup (see Section 9.2.7.4.10 on p. 1854),
- ▼ MP measurement unit selection (see Section 9.2.7.4.15 on p. 1858),
- ▼ setup of the material, material's density and hatch parameters of a part that are used by default in a new document (see Section 9.2.7.4.1 on p. 1843).

2.14.6.6. Hatch parameters of a model

Hatch parameters of a model can be set up using one of the following commands: **Model properties**, **Body properties**, or **Component properties**.

You can set up hatch parameters of a model's material (style, color, and hatch angle) in the **Material** bar of the **MCC Parameters** tab. These parameters are saved in the model and used when it is displayed on cut/section in the drawing's associative view.



You can set up hatch parameters, if MCC parameters are calculated by density or by mass. If MCC parameters are specified based on a source (available for components and bodies), hatch parameters will also be specified on the source's basis.

In order to specify the hatch style, click the cell of the **Hatch Style** value. A list will be displayed from which you can choose the hatch style, including a custom one (Fig. 2.14.14).

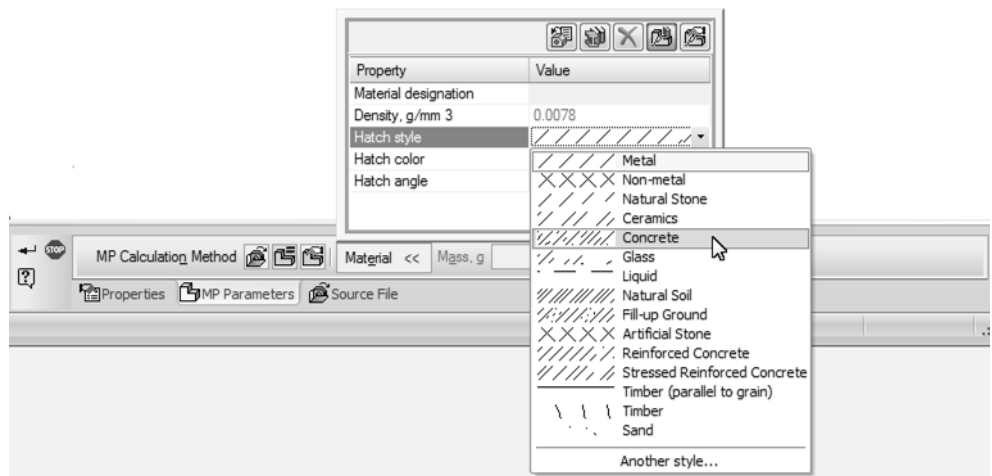


Fig. 2.14.14. Selecting the hatch style

See Section 10.1.1.2.2 on p. 1912 for more information about creating custom hatch styles and saving them in the library.



Hatch style is automatically applied to model if material is selected from the file of densities by clicking the **Select material from list** button (see Section 2.14.6.1.1 on p. 802). This is due to the fact that the file of densities contains information about relation of a hatch style to a certain material. See Section 9.5.5.2 on p. 1897 for more information about preparation of the reference file of densities.

In order to change the hatch color or set its angle, click one of the following cells, respectively: **Hatch color** or **Hatch angle**. The color selection dialog or the list of angle values will be displayed. If necessary, enter the angle value manually.

The specified inclination angle will be applied to hatch on the drawing, if the chosen style supports angle modification.



You can change hatch parameters when setting up material properties to components and bodies in the Property Editor (see Section 5.1.4.2 on p. 1389).



After completing hatch parameters setup click **Create Object** button on the special control panel.

2.15. Service Functions

2.15.1. Model diagnostics

2.15.1.1. Error Warnings

If such model editing was done that invalidated some of its objects, considering the hierarchy and/or parametric relations, then KOMPAS-3D will output the respective diagnostics message. It indicates the specific reason for the conflict or loss of the relation between the model components (for example, “*The feature lost the base object*”, “*The base surface is changed*”, “*Empty Sketch*”, “*Self-Intersection of Contour*”, etc.).

At the same time, the model is not rebuilt, but stays in the state in which it was before calling the editing command. An error flag appears in the Design Tree next to the icons of the objects which became nonexistent after the editing.

If there is an error in at least one model object (element, component, match, etc.), an error flag appears not only next to the icon of this object, but also next to the icon of the model (element or assembly). So, an error existing in the model can be noticed even if the Design Tree is minimized.

To find out the reason for the occurred error, select the erroneous object in the Design Tree and call the **What is wrong?** command from the context menu.

A dialog in which the errors are listed, that occurred during this object rebuild will be displayed.

Errors are displayed in the dialog as a structured list. Description of each error is located at the level next to the name of the failing object.

When browsing through all model errors, its failing components (geometry-producing features, auxiliary axes and planes, copies, chamfers, shells, components, mates, etc.) will appear in the list at the level below the model name.

The help system contains recommendations on possible error-fixing techniques.

To get explanation for the specific error and general recommendations on fixing it, select the error description in the dialog and click the **Help** button. Read the displayed page of the Help system, think, which of the described cases is most similar to the error in your model, and remember recommendations on its fixing.

To fix an error, edit a feature, sketch or mate, in which it occurred, or the source elements according to the Help system recommendations. Often, to fix an error, it is enough just to edit the failing model again, reverting to the old object parameters.



If a model has several errors, fix those in the order in which the failing elements appear in the Design Tree.

Some errors make it impossible to maintain elements, continue construction and editing a part. For example, after editing a contour, reference vertices for making construction axes and planes disappeared in the sketch, and a derivative element cannot be rebuilt (the model “fell apart”, it cannot be further used before fixing the error).

Other errors are not so critical and do not prevent from working with the model further. For example, when translating a sketch to another face, the parametric relations of the sketch graphical objects were lost. In this case, graphical objects are not rebuilt in the sketch, the contour formed by them stays the same, and the element based on it is not destroyed, thus allowing to maintain the dependent elements.



In any case, it is recommended to fix an error right after its occurrence.

2.15.1.2. Model Rebuilding Warnings

Upon assembly editing (moving the components, changing the value of the variables, etc.), the links between the model components may be broken. For example, auxiliary elements may remain after their reference objects have been moved. The components that caused such conflicts are marked with red “ticks” in the Design Tree.



To resolve the conflicts, you will need to rebuild and/or move all objects so that their shapes, parameters and positions match the positions of reference objects and comply with the imposed matings. You do this using **View — Rebuild Model** command. For details about links, see Section 2.14.3.7 on p. 795.

Sometimes, the warnings may be displayed after rebuilding a model (exclamation marks in red circles).

2.15.2. Measurements

While working with a model, you may need to find a distance or an angle between the vertices, edges or faces, a curve length, a face area.

KOMPAS-3D supports measurements of various geometrical parameters, as well as calculation of mass-centering properties of a model (volume, mass, center of gravity coordinates, axial and centrifugal moments of inertia, directions of the principal axes of inertia).

Geometrical parameter measurement results can be saved in the model as a variable values.

Measurement commands are grouped in menu **Service**, and buttons to call commands— in panel **Measurements and diagnostics (3D)** (Fig. 2.15.1).



Fig. 2.15.1. **Measurements and Diagnostics (3D)** Toolbar

After call of any of the measurement commands, the Properties bar displays the **Measurement** tab. It contains elements allowing to customize measurement process parameters. These elements are presented in Table 2.15.1.

Tab. 2.15.1. Control elements of measurement parameters

Element	Description
Decimal Places	A list that manages precision of presentation of calculation results. Minimal decimal place is 0, and maximum is 10.
Length Units	The list allows selection of length units: millimeters, centimeters, meters.
Angular Units	The list allowing selection of angle units: degrees, radians. Only for the Distance and Angle command.
Mass Units	The list allows selection of mass units: grams, kilograms. Only for command MP Models .
Brief	Options that allows to get brief or full measurement results. Only for commands Distance and Angle , Model Mass Properties , Deviation of Surfaces .



The buttons to call the **Check for Intersections** and **Object Info** commands are located on the **Measurements and diagnostics (3D)** bar. More information about working with them is provided in Chapters 2.15.3 and 2.15.6, respectively.

You can set up precision to calculate MP and areas of faces in the model. This setup is described in Section 9.2.7.4.10 on p. 1854.

2.15.2.1. Distance and Angle

You can calculate the distance and, if possible, the angle between two specified objects. Such objects can be coordinate and auxiliary axes and planes, faces, edges, vertices, 3D curves and points. Can be selected in any combination (for example, a plane and a vertex, an edge and an axis, two faces).



To calculate the distance and angle between two objects, enable the **Distance and Angle** command.

Click on the first object in the Model window. The selected object is highlighted.

Move cursor to the 2nd object.

The Information window will be displaying the distance between them (if it is not zero) and the angle value between them (if objects are neither parallel nor perpendicular). The measurement results will appear near the cursor.

If objects are intersected or parallel or perpendicular, then the respective message appears near the cursor.

The displayed measurement result depends on types of the specified objects and their relative positions.

Description of results is given in Table 2.15.2.

Left click on the 2nd object to enter measurement results to the Information window.

Window management and its menu commands are described in Section 3.8.1.2.

Objects can be selected before calling the command. After the command is called, the objects are highlighted and the following items appear on the screen: Information window with measurement results, ghosts of all measured distances and rotation axes (if the rotation surface is specified).

Tab. 2.15.2. Measurement Results

Value Designation	Measurement results description
L	Distance between objects. Is measured when it's possible to uniquely determine the distance between objects, e.g. between two vertices. Also the L distance is measured if one of the objects is the finite one (face, edge, curve), and the second object is and infinite one (auxiliary, coordinate axis or plane). In this case the result of the L measurement is the shortest distance between the objects.
L (min), L (max)*	The minimum and maximum distance between objects. Is measured between the points of two objects that are the most proximate L (min) and the most distant L(max) from each other. If the objects intersect each other the L (min) value does not appear on the screen. The L (min) and L (max) measurement is made in case when both objects are finite. One of the objects can be pointwise.
L (norm)	Distance between parallel planes. Is measured when the specified objects are situated in parallel planes.
A	Angle depending on the specified objects type: <ul style="list-style-type: none"> ▼ for the rectilinear objects— the angle between crossing or intersecting straights containing these objects, ▼ for a rectilinear object and a plain object— angle between the rectilinear object and its projection on the plain object, ▼ for two plain objects— angle between the planes of the objects, ▼ for a rectilinear object and a rotary surface— angle between the object and the axis of rotary surface, ▼ for a plain object and a rotary surface— angle between the plane of the object and the axis of rotary surface; ▼ for two rotary surfaces— angle between the axes of rotation.
Complement Angle*	An angle which completes the measured angle A to 180° or 360°.
Distance to the axis	Distance from the point to the axis of a rotary surface. Is measured if one of specified objects is a rotary surface.

Tab. 2.15.2. Measurement Results

Value Designation	Measurement results description
Distance between axes	Distance between the axes of rotary surfaces. Is measured if both specified objects are the rotary surfaces.
In projections on the absolute coordinate system axis*	dx, dy, dz — distance projections (L , L(min) , L(max) , L(norm) , Distance to the axis or Distance between axes) in millimeters on the axis of the absolute coordinate system X, Y, Z.

* This parameter is calculated if option **Brief** is disabled on the Properties bar.

To obtain additional measurement results, deactivate the **Brief** option on the Property bar and repeat object selection.



Remember Status button on the Special Control Bar allows to remember first selected object. For example, if you need to measure distance and angle between one object and several others, click the **Remember Status** button after first object selection and then specify other objects.

2.15.2.1.1. Measurement object

When using commands for measuring distance, angle, face area or edge length, you can create **measurement object**. Meanwhile, information variables with values similar to the measurement results and related to the object are automatically generated in the model. For details about informational variable see Section 7.1.1.2.1 on p. 1570.

To create measurement object, specify object (objects) by clicking the mouse in the model window and then click the **Create Measurement** button on the Property bar.

Measurement object corresponding to the last performed measurement appears in the Design Tree and in the Variables Window.

When selecting measurements in the Design Tree, this measurement object (objects) are highlighted in the model window.



When using the **Measure Distance and Angle** command, depending on the **Brief** option status, a measurement object with full or shortened set of variables may be obtained.

The measurement object cannot be edited— it can only be deleted from the model or exclude from calculations.

Each measurement preserves the links with its objects, i.e. once you make changes which influence the measurement results in the model, a variable matching this measurement is updated. If you exclude the measurement object from the calculation, a new value of this variable will be calculated once you include this object in the calculation.

In some cases, editing a model may cause degeneration of the measurement object. For example, when two points are overlapped and have zero distance. In this case, the measurement object is marked as corrupted both in the Design Tree and the Variables window.

If you remove an object used in measurement, then the measurement object is also removed.

2.15.2.2. Edge Length



To calculate the 3D curve length or the part face perimeter, enable the **Edge Length** command.

To measure an object length, move cursor to such object. The measurement results will appear near the cursor.

Left click on the selected object to enter the measurement results to the Information window.

Message window with the measurement results will be displayed. The selected object is highlighted.

Window management and its menu commands are described in Section 3.8.1.2.

You can view lengths of other objects by moving cursor to them. To enter measurement results to the Information window click sequentially on these objects.

The selected objects are highlighted. The Message window will display the list of measured lengths. The sum of the measured values will appear at the end of the list.

Select multisegment curve in Design Tree to measure its full length.



If you need to measure a face perimeter, select that face right after calling the command. In that case, the sum of measured values will represent its perimeter.

Reselect an object to exclude it from the list. Highlighting will be removed, the record on its length will be deleted from the window, and the sum of lengths— calculated again.

Objects can be selected before calling the command. After calling the command, Information window with measurement results appears on the screen.

Depending on the measurement results, you can create measurement objects (see Section 2.15.2.1.1 on p. 815).

2.15.2.3. Area



To measure the area of part faces, call command **Area**.

To measure a face area, move cursor to such face. The measurement results will appear near the cursor.

Left click on the selected face to enter the measurement results to the Information window.

Message window with the measurement results will be displayed. The selected face is highlighted.

Window management and its menu commands are described in Section 3.8.1.2.

You can view areas of the other faces by moving cursor to them. To enter measurement results to the Information window click sequentially on these faces.

The selected faces are highlighted. The Message window will display the list of measured areas. The sum of the measured values will appear at the end of the list.

To exclude a face from the list, indicate it again. Highlighting will be removed from this face, the record on its area will be deleted from the window, and the sum of areas— calculated again.

Objects can be selected before calling the command. After calling the command, Information window with measurement results appears on the screen.

Depending on the measurement results, you can create measurement objects (see Section 2.15.2.1.1 on p. 815).

2.15.2.4. Model Mass Properties



To calculate mass properties for the current model, call the **Model's MP** command.

By default, brief model information will be displayed in the Information window. These are the following data:

- ▼ Model surface area
- ▼ Model volume
- ▼ Model mass
- ▼ Center of mass coordinates according to the absolute model coordinates.

To see additional info, deactivate the **Brief** option on the Property bar. The following data will be displayed in the Message window:

- ▼ Axial moments of inertia (in absolute, central and world coordinate systems)
- ▼ Products of inertia (in absolute and central coordinate systems)
- ▼ direction of principal axes of inertia.

The **Mass Center** button on the Property Bar controls displaying model's mass center. If the button is enabled, the ghost of the main central system of coordinates. Axes of this system of coordinates are designated as J1, J2, and J3 (main inertia moments in the descending order of their values).

To create a point in the center of mass, activate the **Point** option on the Property bar, and then click the **Center of Gravity** button. A point called as "Point in CG" will be created in the model.



The **Point in CG** will not change its position when editing model in future. To create a new point corresponding to actual mass center, use the **Model Mass Properties** once again.

2.15.2.5. Deviation of Surfaces

You can measure distance between two objects. Such objects can be coordinate and auxiliary planes, faces, surface objects.

Further, the **surface** term will be used for measurement objects designation.



To measure the distance between the surfaces, call the **Deviation of Surfaces** command. Then specify surfaces one by one in the model window. Surfaces can be specified prior to calling the command.

Distances are measured between points of the first surface and their projections to the second surface in the specified direction.

The number of points is defined by the user, their location on the surface is defined automatically. Direction of measurement depends on the selected method of measurement:

- ▼ normally to the first surface,
- ▼ parallel to the selected guide object.

You can choose the measuring method using the switches from the **Method** group.



Switches of the **Direction** group are used to choose the direction of measurement: direct, reversed, or in both directions. In the Model window, the direction of measurement is shown by the ghost. It should match the direction to the second surface, otherwise the measurement will fail.

Both measurement directions depend on the surfaces' form and relative position. For example, let the measured surfaces sectioned by plane have the following view shown in Fig.2.15.2 Measurement is made using the guide object. If you make measures only in one direction, then a part of distances will not be measured; the figure illustrates the lines of these measurement by intermittent points. To measure all distances between the surfaces, select option **Two Directions**.

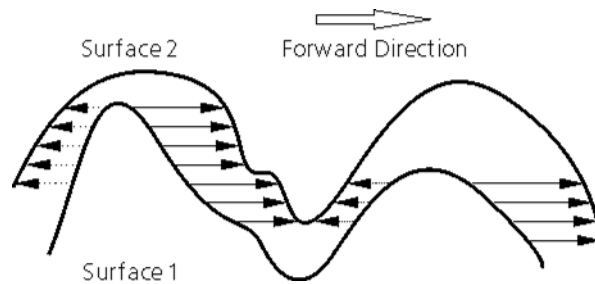


Fig. 2.15.2. Section of Surfaces

The user should make the decision about the necessity of performing measurements in both directions.

Fields with the **Points in U** and **Points in V** counter serve to set the number of points of measurement for **U** and **V** parameters. By default, field values are equal to 20. The total number of measurements is equal to the product of the fields' values.

Fields with the **Minimums** and **Maximums** counter serve to set the number of the first sought minimums and maximums the parameters of which are displayed in the Info window. By default the value of these fields is 1.

The **Briefly** option serves to choose the number of measures, parameters of which will be shown in the Information window as a result of the command execution. If this option is enabled, only the first sought maximums and minimums will be displayed. If the option is disabled, all the results of measurements will be displayed.

The measurement results are shown in the Information window.

Window management and its menu commands are described in Section 3.8.1.2.

It contains the following parameters:

- ▼ Number of measurements,



Number of measurements does not necessarily coincide with the specified number of points. E.g. the measurement is not performed if the measuring line does not intersect the second surface.

- ▼ Minimum distance between surfaces in the points of measurement,

- ▼ Maximum distance between surfaces in the points of measurement,
- ▼ Simple average of distances in the points of measurement,
- ▼ Absolute minimum of distance between surfaces,



The measuring point of the absolute minimum does not necessarily coincide with the points of the defined set.

- ▼ Specified number of measurement results that are the closest to minimum distance (the first sought minimum values),
- ▼ Specified number of measurement results that are the closest to maximum distance (the first sought maximum values).

For each measurement from the first sought minima and maxima the following data set is displayed (see Fig.2.15.3):

- ▼ measured distance,
- ▼ point of measurement coordinates on the first and the second surfaces,
- ▼ arithmetical average distance.

If you measure the distance in both directions, a measurement direction arrow appears in the beginning of each row with measurement results. Direct measurement is marked with the error pointed to the left, reverse measurement— error pointed to the right.

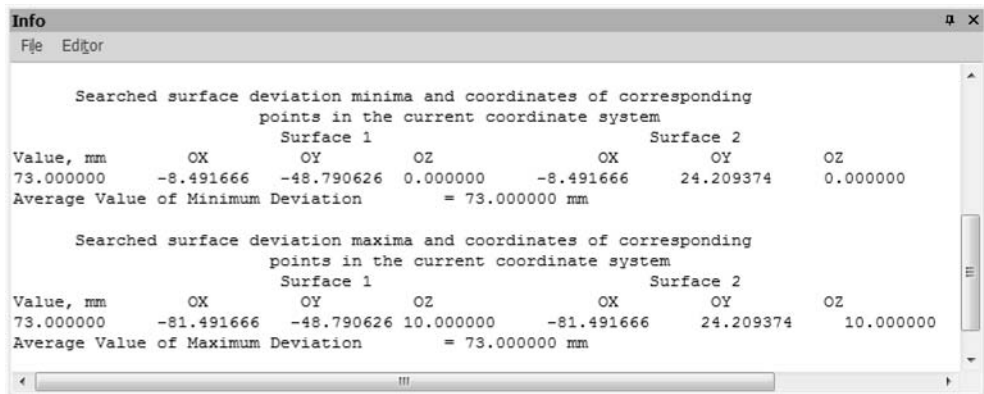


Fig. 2.15.3. The first minimum and maximum measurement results in the Information window

If while running the command, the display of full set of measurement results is selected, the coordinates of measurement points and the measured distance will be displayed for all measurements.



If during the current command session a remeasurement is made, the results of the previous measurement shown in the Information window will be replaced by the new ones without saving. In order not to lose the measurement results it's necessary to save them into the file on disk using the commands of the Information window's menu.

2.15.2.5.1. Measuring Normal to Surface 1



In order to make the measurement it's necessary to click the **Normal to Surface 1** switch.

In the model document window, select the surface to measure the distance from. The object name will be displayed in the **Surface 1** field.

Select the second surface. The object name will be displayed in the **Surface 2** field.



Coordinate or auxiliary plane may be selected as the second surface.

Select the direction of measurement so that the arrow ghost is directed from the first surface to the second. If the measuring direction is chosen incorrectly, an attempt of the command executing will result in the display of the message *Calculation parameters are set incorrectly*.

Define the number of measurements for the **U** and **V** parameters as well as the number of the displayed minima and maxima of the measured values.

Using the **Briefly** option define the range of measurements results displayed in the Information window.



To perform measurements, click the **Create Object** button on the Properties bar.

The measurement results will be shown in the Information window. In the model window the measuring lines connecting the measuring points of the first sought minima and maxima will be shown. Measurement lines of the first sought minimums values are shown with solid lines, and the first sought maximums values — with dotted lines.



The system will wait for changing of the command parameters for further measurements.

The **Interrupt Command** button is used to stop execution of the command.

2.15.2.5.2. Measurement using the guide object



To perform a measurement operation click the **Guide Object** switch.

In the model document window, select the surface to measure the distance from. The object name will be displayed in the **Surface 1** field.

Select the second surface. The object name will be displayed in the **Surface 2** field.



Coordinate or auxiliary plane may be selected as the second surface.

Select a guide object. The following objects can be selected as guide objects:

- ▼ Straight objects: axis, line segment, straight edge, polyline segment. When selecting a straight object, measurements will be done parallel to it.
- ▼ Flat: flat face, coordinate or auxiliary plane. When selecting a flat object, measurements will be done perpendicular to it.



If there is no required guiding object in the Model, you may construct a direction vector (see Section 2.8.5 on p. 613).

Select the direction of measurement so that the arrow ghost is directed from the first surface to the second. If the measuring direction is chosen incorrectly, an attempt of the command executing will result in the display of the message *Calculation parameters are set incorrectly*.

Define the number of measurements for the **U** and **V** parameters as well as the number of the displayed minima and maxima of the measured values.

Using the **Briefly** option define the range of measurements results displayed in the Information window.



To perform measurements, click the **Create Object** button on the Properties bar.

The measurement results will be shown in the Information window. In the model window the measuring lines connecting the measuring points of the first sought minima and maxima will be shown. Measurement lines of the first sought minimums values are shown with solid lines, and the first sought maximums values — with dotted lines.



The system will wait for changing of the command parameters for further measurements.

The **Interrupt Command** button is used to stop execution of the command.

2.15.3. Intersection Detection

Sometimes, one of the purposes of assembly model is to find out the locations of unwanted intersections and tangencies between components (“stumbings” in a design).



To detect such places, click the **Intersection Detection** button on the **Measurements and diagnostics (3D)** toolbar or select its name from the **Service** menu.

After the command is called specify one by one the components or the bodies, that have to be checked for intersection. The selected objects will be shown in the **List of Objects** on the Properties bar.

- ▼ If components are selected for checking, then the intersections will be found between the bodies which belong to these components (not between the bodies that belong to one component). Components are selected in the Design Tree.
- ▼ If bodies are selected for checking, then the intersections will be found between them. Bodies can be selected both in the Model Tree and the Model window.

If you also need to identify tangencies of components of bodies, enable the option **Consider Tangencies as Intersections**.



After selection of components click the **Detect Intersections** button on the Advanced Control Panel.

In the **Intersections Detected list** on the Property Bar, the serial number of intersection and the names of intersecting (tangential) objects will appear. The model window will display an area frame of their intersection and/or tangency line.



To escape the intersection detection command, press <Esc> key or click the **Interrupt Command** button.

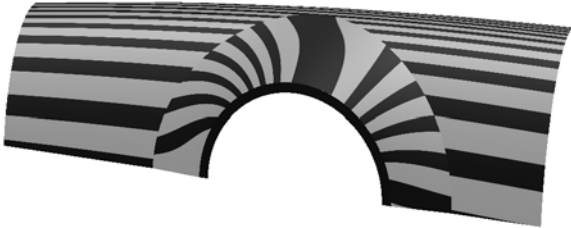
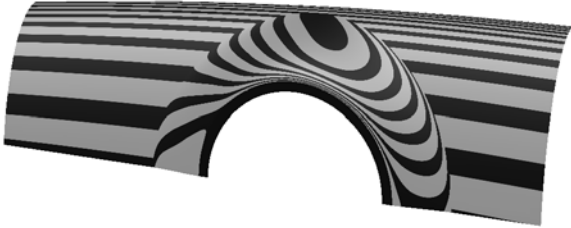
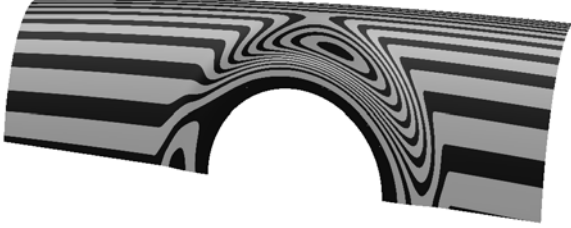
2.15.4. Check the model surfaces for smoothness

When modeling parts (for example, box-type parts), it may be necessary to check surfaces for smoothness thoroughly. For the visual assessment of model surface smoothness, a special mode to display body is used.

In this mode, all faces of the model are represented as mirroring the environment, for example, color bars on the spherical surface. You can visually evaluate the smoothness of surface edges' joints (see Table 2.15.3), and identify subtle curvature changes inside the edges (see Fig. 2.15.4).

Enabling the special model display mode is described in Section 2.15.4.1; information about display settings can be found in Section 2.15.4.2.

Tab. 2.15.3. Assessment of smoothness of the faces' intersections using "Zebra"

Face mating condition	The form and arrangement of the "zebra's" stripes meeting the mating condition
Not specified	Stripes do not adjoin on edges: 
By Tangent	Stripes on edges adjoin with a bend: 
Smooth	Stripes on edges adjoin smoothly: 

2.15.4.1. Enabling model smoothness check mode



In order to enable the model smoothness check mode, call the **Smoothness check mode** on the **Modes** bar or select its name in the **Service — Smoothness check** menu. After calling it, model appearance changes. By default, the **Zebra** method of display is used.



Fig. 2.15.4. Check surface for smoothness
(the surface is a little convex):

- a) dimmed wireframe display of the model— the convex is practically invisible;
b) model's smoothness check mode (**Zebra** method) – the convex can be clearly seen

When working with the document, you can set the model display parameters (for example, change the width of “zebra's” stripes or choose another method of display— **Reflection of the environment**). In this case, model appearance corresponds the settings made.

Model display methods and setup of corresponding them parameters are described in Section 2.15.4.2.



The smoothness check mode usage causes an extra load on the memory and operating system resources resulting in a slow-down of the KOMPAS-3D system.

2.15.4.2. Display parameters setup

Smoothness check mode parameters can be customized. The setting is accessible if there is at least one surface in the Model.



To perform setup, call the **Smoothness check parameters** command by one of the following methods:

- ▼ by choosing from the menu of the **Smoothness check mode** button on the **Modes** bar.
- ▼ by choosing from the **Service — Smoothness check**.

Model appearance will change (if model smoothness check mode is not enabled in the document). The display parameters management interface items will appear on the Property bar.

Two model display methods are possible:

- ▼ **Zebra** is a sphere with stripes is reflected on the model's edges (this method is used by default),
- ▼ **Environment reflection** is a sphere with the bitmap image from file applied to it is reflected on the model's edges.

In this case, model is located inside an imaginary sphere.

Model display method is selected from the same-name drop-down list.

For **Zebra** method, set the color and display parameters of stripes (see Section 2.15.4.2.1).

For **Environment reflection** method, select the file with the required image and set its display parameters (see Section 2.15.4.2.2 on p. 825).

When changing model display parameters, its appearance automatically changes in the window.

If the boundaries of the displayed stripes or sections of the selected image have irregularities, enable the **Enhanced drawing precision** option. The boundaries will be more regular.



To finish the command, click the **Interrupt Command** button on the Special Control Panel. If the smoothness check mode is not enabled in the document, the model will take its previous shape. If this mode is enabled, the model display will correspond the settings made.



Model display parameter settings remain effective till the end of the current document working session.

2.15.4.2.1. Zebra display method



By default, black stripes on white background are used for this method of displaying the model. You can customize band parameters and change their color using the control elements of the Properties bar (Fig.2.15.5). Background color cannot be changed.

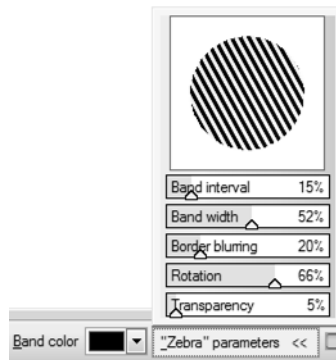


Fig. 2.15.5. Color and stripes parameters control elements

Stripe Color field serves to change color of the stripes. To select the required color, you need to click on the triangle in the right part of the field and specify color in the color palette appeared on the screen.

To set the parameters of the stripes, use the **Zebra parameters** bar. The bar contains preview area and fields corresponding to the following parameters:

- ▼ frequency of the stripes,
- ▼ width of the stripes,
- ▼ boundary feathering,
- ▼ rotation,
- ▼ transparency.

The slider in each field serves to control corresponding parameter value. The preview pane is used for visual assessment of the changes made— it reflects the sphere with stripes on it. When changing color or parameter values, the image in the preview area automatically changes.

2.15.4.2.2. Environment Reflection method



To get a model image by this method, an image, which is contained in a BMP, GIF, JPEG, PNG, TIFF or TGA format bitmap file, is used. Using property bar control elements (Pic. 2.15.6) you can choose the required file and set parameters of the image superimposing on the reflected sphere.



Fig. 2.15.6. Image selection elements and setting parameters of its superimposing on the sphere's surface



To select the file, you need to click **Image File** button and specify the required file in the standard file selection dialog. The full name of the file shall appear in the **Source File** field. The **Reflection parameters** bar will become available to set the parameters of the selected image superimposition on the surface of the reflected sphere.

The bar contains preview area and fields corresponding to the following parameters:

- ▼ horizontal shift,
- ▼ horizontal scale,
- ▼ vertical shift,
- ▼ vertical scale,
- ▼ rotation,
- ▼ transparency.

The slider in each field serves to control corresponding parameter value. The preview pane is used for visual assessment of the changes made— it reflects the sphere with the image on it. When selecting other image file or parameter values, the image in the preview area automatically changes.



Path to the folder with images (i.e. the folder available by clicking **Image File**) can be set in the PICTURES environment variable KOMPAS-3D (see Section 9.5.4 on p. 1886). By default, this folder is *Samples\Surfaces\Maps* relative to the main system installation folder. If you want to change the default path to the folder with images, then use the *Kompas.ini* file (see Section 9.5.3 on p. 1883).

2.15.5. Model Section

During modeling a model, you may find it difficult to assess whether the parts and nodes of the assembly model are placed correctly and whether the construction items of a part and thickness of its walls are appropriate. In this case, use the model section mode for visual control purposes.

When this mode is on, the model is displayed as a trimmed place or a set of planes. The position of the cutting plane (or planes) can be set when creating the model section (see Section 2.15.5.2 on p. 828).

You can create multiple section in a model. One of them is active.



To enable the model section mode, choose **View— Model Section — Display Section** or click **Model Section** on the **View** panel.

If no sections are available in the model, then choosing the command will launch the section creation process. If there are any sections in the model, the model's appearance will be updated. The model will be displayed trimmed by the current section and the model section mode icon will appear in the top right corner of the model window.

You can make another section a current one. To do this, select in from the drop-down menu of the **Model Section** button. The model appearance in the window will be automatically updated.

The section mode is also enabled once the section is created. The created section becomes the current one.

You can perform any operations of object construction, modification, enable other modes (for example, the context editing mode, the sketch mode, etc.) in the model section mode.

When operations are performed, model vertices, edges and faces, including trimmed ones, can be clicked unlike section vertices, edges and faces.

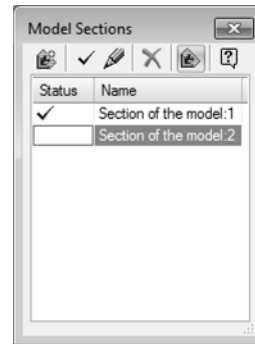
To finish work in this mode, go to **View — Model section — Display section** again, click **Model section** in the **View** bar again or click the mode icon in the model window.

To create a set of sections in the model, use the **Model Sections** dialog described in 2.15.5.1. In this dialog, you can create, remove or change the sections as required.

2.15.5.1. Section Management



To create a set of section, use the **Model Sections** dialog (Fig. 2.15.7). To access the dialog, choose **Section Management...** command from the **View — Model Section** menu or from the drop-down menu of the **Model Section** button on the **View** panel.

Fig. 2.15.7. The **Model Section** dialog

The dialog provides a list of sections created in the model as well as controls for managing the sections.

The list of properties has the table form. It allows you to rename sections or change their statuses.

To rename a section, double-click the corresponding cell in the table and type the name you want.



If you want to make the section a current one, choose the row of this section in the table and click **Current Section** on the dialog toolbar or double-click the *Status* cell corresponding to this section. When the section mode is on, the model appearance in the window will be automatically updated.



If the section mode is off, you can enable it without closing the dialog. To do this, click **Model Section Mode**. Clicking the button again will disable this mode. When enabling/disabling the mode, the dialog is displayed on the screen.



To create a new section, click the **Create section** button and perform the steps described in Section 2.15.5.2.



Using the **Edit Section** button, you can change the section parameters (see Section 2.15.5.3 on p. 833).



Once you click the section creation or editing buttons, the **Model Section** dialog will be closed.



To delete an object, select it in the list and click the **Delete** button. If this section is current, the following section in the list becomes current after its deleting.

After performing all necessary operations in the dialog, close it.



Actions performed in the dialog cannot be undone.

2.15.5.2. Creating a Section

A model section can contain multiple cutting steps. You can configure individual parameters for each step. By default, a section contains only one step. You can add the required number of steps when creating or editing a section.

On each step, a certain fragment of the model is trimmed by one or more planes. The number of planes depends on the mode you've previously chosen for setting the step parameters.

You can create as many cutting steps as required, however each model section cannot contain more than 6 section planes.

You can launch the section creation process in the following ways:



- ▼ by calling the **View — Model Section — Display Section** command or clicking the **Model Section** button on the **View** panel when no sections have been created in the model yet,



- ▼ by clicking the **Create Section** button in the **Model Sections** dialog.

The model window will then display a ghost of the section plane, and the Properties panel will display the **Parameters** (Fig. 2.15.8) and **Properties** tabs.

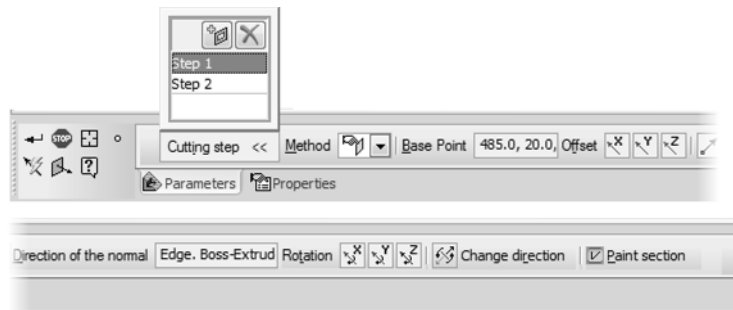


Fig. 2.15.8. Creating a model section, the **Parameters** tab

The section plane ghost is placed alongside the XY plane of the absolute coordinate system. This location of the plane corresponds to the first cutting step. You can also set another parameters of the first step as well as add additional steps as required.

The list of cutting steps is available on the **Cutting Step** panel of the **Parameters** tab. By default, it contains only one step.

When the parameters you've provided are insufficient for creating a cutting step, the system will display the (!) signs to the right of the step name in the list.

Specify the required parameters for the first cutting step. A set of parameters depends on how you choose to place the section plane in the **Method** drop-down list. The following methods are available:



- ▼ **Free Placement** (see Section 2.15.5.2.1),



- ▼ **Offset Plane** (see Section 2.15.5.2.2 on p. 831),



- ▼ **Rotated Plane** (see Section 2.15.5.2.3 on p. 832),



- ▼ **Outline Box** (see Section 2.15.5.2.4 on p. 833),



- ▼ **Zone** (see Section 2.15.5.2.5 on p. 833).

In some cases, objects available in the model are used when setting cutting step parameters. For example, when you place a section plane using the **Rotated Plane** method, the base plane and the revolution axis are placed relative to the objects you've selected. To change the object, click the **Reselect** button on the Advanced Control panel.



When **Free placement**, **Offset plane** or **Rotated plane** is used for positioning the section plane, you can manage the cutting direction. In case of free placement of a plane, the cutting direction is shown by Z axis of the positioning element, in case of plane offset or rotation — the blue arrow. To change the face extension direction click the **Change Direction** switch.



When using these placement methods, you can also enable the model orientation, so that the section plane is parallel to the screen. To do this, click **View Along the Normal of the Section Plane** button on the Advanced Controls Panel.



For the **Offset Plane** and **Rotated Plane** methods, the **Change Direction** and **View Along the Normal of the Section Plane** will become available once you choose the base objects.



If you want to add multiple cutting steps in the section, create a new step by clicking the **Add Cutting Step** button on the **Cutting Step** panel. A new step will appear in the list. Choose a building method and specify the placement parameters for the section plane as described above.



To remove a cutting step, select it in the list and click **Delete**.

To display one or more ghost edges on the place where the created plane intersects the model, enable the **Fill Section** option. A color and optical properties of this edge can be set on the **Properties** tab (see below). The ghost face is not an object and is displayed only when the section mode is enabled.

To hide the ghost edges in the mode, disable this option. This can help you to visually assess whether the components of the assembly model are placed correctly (e.g. whether any components intersect each other).



The **Paint section** option is enabled/disabled for the entire newly-created section rather than its individual steps, i.e. ghost faces are displayed (or not displayed) for all cutting steps.



You cannot use any ghost edges as well as its edges and vertices for performing any operations in the model. They are not able to be selected.

To assign a name to the new section, expand the **Properties** tab and type the name you want in the **Name** field.

You can also set the color and optical properties of the ghost edge on this tab. To do this, use the **Set Color** group of controls. The following options of a flat object setting are available:



▼ **By the source object**,



▼ **Manually**.

When setting a color manually, you can also configure the optical properties. For details on how to set the color and optical properties, see Section 2.14.2 on p. 784.



In order to finish creating the report, click the **Create Object** button. The newly created section becomes the current one. The model appearance is updated. The model is now trimmed with the planes of this section, i.e. the model section mode is enabled.

2.15.5.2.1. Free Placement of a Section Plane



To activate the free placement of a section plane, you can use the controls on the Properties panel (see Fig. 2.15.8 on p. 828) or the special positioning control.

Placement of a plane includes setting the position of the reference point which defines the position of the plane in the model and setting the orientation of the plane.

The model window will display the special positioning control which overlaps with the reference point. It is a kind of a coordinate system consisting of three large arrows of red, green and blue colors which correspond to the positive directions of the X, Y, and Z axes. The planes perpendicular to the arrows display the arcs of the corresponding colors. The center point of the control is marked with a sphere or a cube (the cube is displayed when the reference point is associated with a model object). The XY plane of the control overlaps with the section plane you are creating. The positive direction of the Z axis shows the cutting direction.

You can change the position of the reference points for the section plane by moving this control. Rotating the control will change the orientation of the plane in the model. For details on how to move and rotate the control, see Section 2.11.2.5.3 on p. 696.



If a reference point of the section plane is associated with an object (the center point of the control is marked with the cube), you cannot move the control arbitrarily. However, you can delete the association by moving the control along the axis.

You can also use one of the following methods to set the position of the reference point.

- ▼ Selecting a point object in the Design Tree or the Model Window. In this case, the reference point will be associated with this object.
- ▼ Set the offsets along the coordinate axes. To move the plane along the required axis, activate the switch for this axis in the **Offset** group:
 - ▼ **Offset by X axis,**
 - ▼ **Offset by Y axis,**
 - ▼ **Offset by Z axis.**



The **Offset** field will appear on the Properties bar. In this field, enter the distance that the plane should be set off for.



- ▼ Point creation. This will create an association. Click the **Create Object** button on the Special Control Panel. This will initiate the process of point creation (see Section 2.5.2.11.1 on p. 347). Select a construction method and specify the required parameters. These parameters will define the position of the reference point. Note that in this case the point building process is used only for defining the position of the reference point. No new point is created in the process. After the point is created, the system will proceed with creating the plane.

The Properties panel provides the **Reference Point** field. This field is for information only (you cannot change its value). Depending on the positioning method used for the reference point, the field can display either coordinates of the reference point or the name of the object it is as-

sociated with. The content of this field changes automatically when the position of the reference point changes.

You can use the following methods for setting the orientation of the plane.

- ▼ Specify an object which defines the normal to the section plane (the normal overlaps with the Z axis of the positioning control). The object is specified on the Design Tree or in the model window. Objects of the following types are available:
 - ▼ straight objects — the normal is parallel to the object,
 - ▼ flat object (including a sketch and a flat curve) — the normal is perpendicular to the object plane,
 - ▼ revolution surface — the normal is parallel to the revolution axis.



You can choose a sphere only when it owns the reference point of the plane. In this case, the normal overlaps the perpendicular line to the sphere in the reference point.

- ▼ Setting the section plane rotation angle about the axis of the positioning element.



In group **Units make** active one of switches:



- ▼ **About X axis,**



- ▼ **About Y axis,**

- ▼ **About Z axis.**

The **Angle** field will appear on the Properties bar. Enter the required value of the angle in this field. The section plane ghost will rotate about the selected axis to the entered angle.

- ▼ Vector Drawing



Click the **Create Object** button on the Advanced Control Panel. The vector creation process will be launched (see Section 2.8.5 on p. 613). Select a construction method and specify the required parameters. The model will display a ghost of a vector with the specified direction. Finish building the vector. The system will resume the object construction process. The section plane's normal will be parallel to the vector and will have the same direction.

The Properties panel provides the **Direction of normal** field. This field is for information only (you cannot change its value). Depending on the direction setting method for the section plane's normal, the field will display either the guiding object name or the coordinates of the normal vector. When the normal is defined by a vector, the word *Vector* is also shown in this field.

When you change the orientation of the plane, this field is automatically updated.

2.15.5.2.2. Offset Section Plane



Once you select the **Offset Plane** positioning method, the Properties panel will display the controls shown in Fig. 2.15.9.

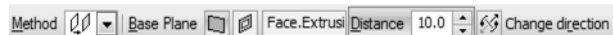


Fig. 2.15.9. **Offset plane** Method

To set up inscription location parameters, do the following.

1. In the Design Tree or in the Model Window, specify a flat object (a plane or a flat edge) which will then be used as a base plane.

The **Base Plane** will display the name of the selected object and the model window will show the section plane ghost overlapping the base plane. The flat object you've selected is highlighted. The cutting direction is shown with a blue arrow.

2. Specify the offset of the section plane relative to the base plane.

To do this, enter the value you want in the **Distance** field. The plane will then be shifted to the distance you've specified.

You can also shift the plane manually. To do this, move the blue arrow which denotes the cutting direction. The following methods are available:

- ▼ arbitrary distance — drag the cursor,
- ▼ exact distance — click the arrow and type the offset value in the field that appears.

When shifting the plane manually, the **Distance** field is automatically updated.

2.15.5.2.3. Reversed Section Plane



Once you select the **Rotated Plane** positioning method, the Properties panel will display the controls shown in Fig. 2.15.10.



Fig. 2.15.10. The **Rotated Plane**< method

To set up inscription location parameters, do the following.

1. Select a flat object (plane or flat face) which will be used as a base plane in the Design Tree or in the Model window. The object you've selected is highlighted in the model window and its name is displayed in the **Base Plane** field.
2. Specify an axis around which you want to rotate the section plane. To do this, choose a linear object (an edge or an axis) on the base plane or on any plane parallel to the base one either in the Design Tree or in the model window.

The **Axis** field will display the name of the object you've specified and the model window will display the ghost of the section plane. It goes through the selected axis parallel to the base plane (or overlaps with it). The cutting direction is denoted by the blue arrow while the rotation plane is denoted by the arc.

3. Specify the rotation angle around the axis you've chosen.

To do this, enter the inclination angle value in the **Angle** field.

You can also rotate the plane manually. To do this, use the arc which denotes the rotation plane. The following methods are available:

- ▼ arbitrary angle — drag the arc with the cursor,
- ▼ exact distance — click the arc and type the rotation angle value in the field that appears.

When rotating the plane manually, the **Angle** field is automatically updated.

2.15.5.2.4. Box-like Section



A model section can be box-like. The part of the model that is outside the parallelepiped is cut. To create such a section, use the **Box** method. When you select this method, the Properties panel will display the controls shown in Fig. 2.15.11.

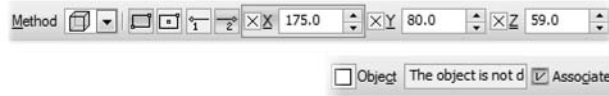


Fig. 2.15.11. The **Box** method

In order to set parameters of the parallelepiped, do the following.

1. Choose the box definition type. To do this, enable one of the following switches in the **Option** group:



▼ **By two vertices,**



▼ **By center and vertex,**

2. Position parallelepiped points.



The position of the first point is set when the **Point 1** switch is active and that of the second one — when the **Point 2** switch is active.



Point coordinates can be input from a keyboard or defined automatically by positioning the point with the mouse in the model window or selecting a point object. These operations are similar to those described in Section 2.5.2.2 on p. 336.



You can also specify the coordinates using the point construction process which can be launched by clicking the **Point Creation** button on the Advanced Control Panel (see Section 2.5.2.11.1 on p. 347). The parameters set during this process define the position of the box point. No new point is created in the process.

Once you set the coordinates of any point of the box, its ghost will appear in the model window.

2.15.5.2.5. Section by Zone



If the model contains zones (see Section 2.15.14 on p. 853), you can select any of them when setting the parameters of a cutting step. To do this, use the **Zone** method. If you choose a zone, all model segments outside this zone will be cut off.

The right zone is selected in the **Zones** tab of the Model Tree. The zone ghost appears in the model window and the zone name — in the **Zone** field of the Properties bar. A second click on the zone cancels its selection.

2.15.5.3. Editing Section Parameters



When you need to edit the parameters of the model section, choose this section in the **Model Section** dialog and then click the **Edit Section** button (see Section 2.15.5.1 on p. 826).

The model window will display a ghost of the section plane, and the **Parameters** and **Properties** tabs will become available on the Properties bar. These tabs provide the same set of controls as when you create a section.

You can edit the parameters of the cutting step you want. To do this, select it on the **Cutting Step** panel and perform the actions described in Section 2.15.5.2 on p. 828. You can also add or remove the cutting steps.

2.15.6. Receiving Info on Model and its Objects



While working with model, you can get reference data about the current part or assembly, or else any object of the part: edge, face, element, surface, component etc. Click the **Object Info** button on the **Measurements (3D)** bar or select the command from the **Service** menu.

To quickly obtain information on any object, move the cursor to it in the model window. The object's symbol and basic data will appear near the cursor (Figure 2.15.12).

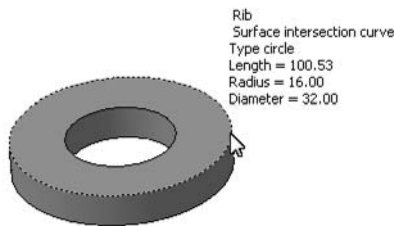


Fig. 2.15.12. Quick Viewing of Edge Info

Left-click the selected object.

The Information window will pop up.

Window management and its menu commands are described in Section 3.8.1.2.

By default, the Message window provides brief information on an object, i.e. only basic data (see Section 2.15.6.1). If necessary, you can also get additional data on the object (see Section 2.15.6.2). For this, disable the **Brief** option on the Property Bar and select the object once more.

The object or objects which information you need to obtain can be also selected before the command is called.

Certain objects can be selected only in Tree, for example, assemblies, parts, elements, sketches, polylines, other— only in the window, for example, edges, faces, vertexes, sketch contour, polylines segments and others— both in Tree and in the window, i.e., central point, auxiliary and construction planes, points, splines.

If objects were selected in the Tree or in the model window before the **Object Information** command was called, the Information window pops up right after the command is called and contains data on those objects.

The Information window provides brief information on an object – basic data (see Section 2.15.6.1).

Sketch should be specified in the Design Tree.

To view information about another object, move the cursor to it. When you click the object, its data will be added to the Information window.

On Property Bar, you can also set the accuracy of calculations, units of length and angle measurement. Change of these parameters is displayed immediately in the Info window.



To finish the command to retrieve info, click **Interrupt Command** button on Advanced Control Panel or close the Info window.

2.15.6.1. Basic Information

Basic information about object:

- ▼ Number of measurement
- ▼ Name of object
- ▼ Object type. Data on type of object are obtained from mathematical description of object. Example:
 - ▼ Edge can be of type *line*, *arc of a circle*, *spline* and others;
 - ▼ Face or surface can be of type *plane*, *toroidal*, *ruled surface*, *spline surface* and other.
 Certain objects, e.g., point or element, do not have a type.
- ▼ Numerical parameters of an object. Set of parameters depends on the object:
 - ▼ For objects, being a kind of a curve line (edge, 3D line, sketch line, etc.), length value is set
 - ▼ For cylindrical, spherical and circular objects (rounding surface, edges and contours as circles/arcs of circle, etc.) radius and diameter values are to be set
 - ▼ For points coordinates are defined
 - ▼ For planes, you should define coordinates of normal vector, etc.

Object parameterization also includes names and values of this object's variables.

For parts and assemblies, values of some system properties (file name, date of creation, and so on; more information about system properties can be found in Section 5.1.3.1 on p. 1380) are specified; for assemblies, general data about the number of first-level components, mates, and elements is specified.

Components of First Level are parts and sub-assemblies being integral part of the current assembly. Components, included into subassemblies (and also subassemblies of subassemblies, etc.), are not components of first level for the current assembly.

- ▼ If the Tree displays the sequence of model construction, the first level components are located on the first level of Tree.
- ▼ If the Tree displays the structure of model, the first level components are located on the first level of section "Components".

2.15.6.2. Additional Information

Additional information on object:

- ▼ Color and visual properties of object
- ▼ Relations in form of lists of initial and derivative objects
- ▼ For faces: parameters of bounding edges, for edges: parameters of final vertices
- ▼ For a variant (i.e. Variant Tree object): numbers of initial and derivative variants.
- ▼ For assemblies: number of components of all levels, i.e., total of components of the first level and other components.

2.15.7. Exploding of Assembly Components

Sometimes, it is necessary to view an assembly in an “exploded” state (so that to see all its components).

Before exploding components, you need to set up the exploding parameters: select components and set the direction and value of their relocation.

To set explosion parameters, call command **Service— Explode Components — Options**.



The same component can be involved in several exploding steps.



One exploding step may involve several components.



A component belonging to a sub-assembly (at any nesting level), can be involved in an explosion step irrespective of other components in the sub-assembly.

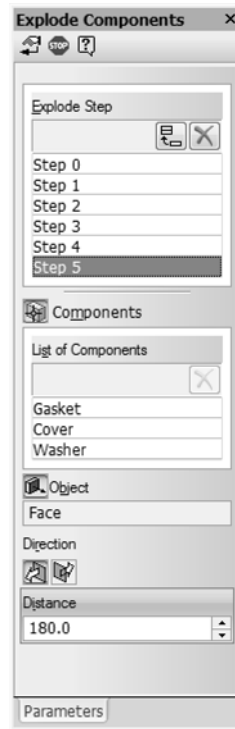


Fig. 2.15.13. Exploding Properties

On Property Bar (Fig.2.15.13) a list of components of explosion steps will pop up.

If the exploding parameters setup of the current assembly was not done yet, the list will be empty.



To add explosion step, click **Add** button.

Then select components, included in explosion step, and parameters of this step:



1. To select components, enable option button **Components** and select the objects required.



2. To define components explosion direction, enable option button **Object**. Components can be exploded in the direction, coinciding with the model's edge (for this, select the required edge in the assembly window) or in the direction, perpendicular to the face (for this, select the required face).

3. Enter the **Distance** in the appropriate field where a component should be moved with respect to its original position.



4. Select direction of displacement of components— **Forward** or **Reverse** by enabling the corresponding option button in group **Direction**.



After setting the components explosion step parameters, click the button **Apply**. The selected components will be exploded according to the specified parameters.

If the resulting exploded state of components is other than expected, edit the exploding parameters.



Similar to that set the required number of explosion steps and tune their parameters.

To delete a step or a component from the list, click button **Delete** in the upper part of the list. Upon completion explosion step setting button **Interrupt Command** on Advanced Control Panel.

After exit from the step setting command, the assembly in the window is displayed in the exploded view mode.



To enable the normal mode of the assembly display, call the command **Service— Explode Components — Explode**. Button for command calling is on panel **Modes**. This command serves to switch between the exploded mode and the normal assembly display. When components are exploded, the button is enabled, and the words “Exploded View” in round brackets are added to the document name in the window title. The document window will display the mode icon notifying that the assembly is currently displayed in the exploded view. To exit the exploding mode, simply click this icon.

When an assembly components are exploded, no assembly editing commands are available (including the **Edit in Window** command), nor the commands for creating 3D curves, surfaces, auxiliary objects, appearance elements, mating commands and commands to work with a BOM.



A subassembly cannot be displayed in assembly window in the exploded state.

2.15.8. 3D Macro Element

A **3D macro element** is a group of model objects. By grouping objects logically into macro elements, you can lay out the Design Tree in a more compact way. This simplifies navigating through the Tree, especially, when working with complex models containing numerous objects.

It is helpful to group objects into macro elements, sharing common function or design purpose.

Objects included in a macro element are displayed in the Design Tree as dependent objects of that macro element and are marked with icons-folders. The procedure of working with them is the same as with objects not included into macro elements: they can be edited, excluded from calculations, deleted etc. Besides, the objects included in a macro element can be used in operations (i.e., copying).

3D macro elements can be nested into each other. The number of nesting levels is unlimited. In other words, you can include into a macro element:

- ▼ several existing macro elements,
- ▼ objects, belonging to any of the existing macro elements.

On Fig. 2.15.14 a macro element *Lug* is shown as an example. It contains the following objects: a lug with outer and inner bosses and a hole, also holes for screws. The bosses and the screw holes are, in their turn, grouped in the same-named macro elements, dependent on the *Lug* macro element.

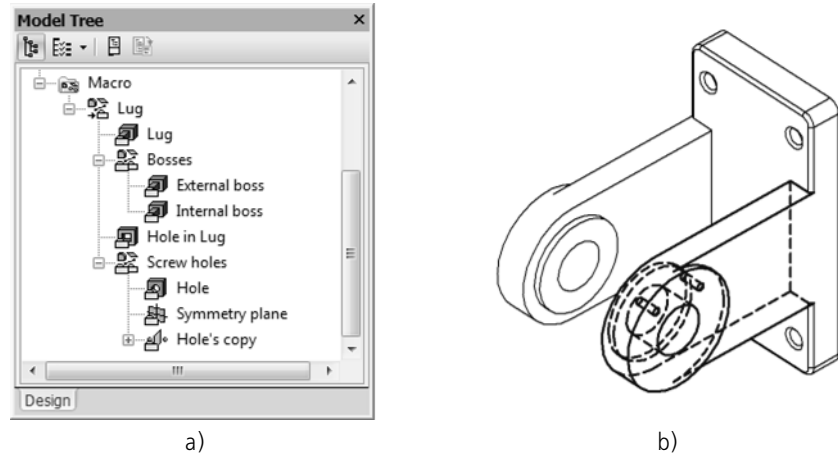


Fig. 2.15.14. Lug Macro element: a) display in the Design Tree, b) model objects, included in the macro element (marked)

By selecting a macro element icon in the Design Tree, all objects included into that macro element are highlighted.

2.15.8.1. Creation of 3D Macro Element

To create a new macro element in the model, call **Service— Create Macro Element** command.



Below the Design Tree macro element shall appear, not containing any object. You can add model objects to it (see Section 2.15.8.3 on p. 840).

To group several existing object in a macro element, select them in the Design Tree and call the **Group into Macro Element** command from the context menu or from the **Service** menu.

A macro element containing indicated object will appear in the position of the bottommost object in included in that macro element. The icons of the objects will be placed at the level following the macro element level. The objects placement order in a macro element is the same as their placement in the Tree before the macro element creating.



If the model design sequence is displayed in the Tree (see section 2.1.2.1 on p. 106), all macro elements are in the *Macro* section.

Pay attention to the fact that macro element can group only such objects that belong to the same “parent” object and are located on the same level, except:

- ▼ Bends in a sheet element (i.e., bends making up a cutting down of)
- ▼ Array instances.

Thus, it is impossible to combine into the macro element the objects, belonging to:

- ▼ Different components of an assembly
- ▼ The macro element and the whole model

- ▼ An assembly component and the whole assembly, etc.

2.15.8.2. Handling the Contents of Macro Element Display

There are two modes of displaying a macro element in the Design Tree:

- ▼ With the contents display
- ▼ Without contents display.

If the contents display is enabled, then the “+” symbol is displayed in the Tree by the macro element icon. Clicking the mouse on this sign (turning it to “-”) opens the list of objects included into the macro element.

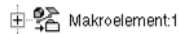


Fig. 2.15.15.

If the contents display is disabled, then the “+” symbol by the macro element icon is missing. You cannot view the contents of such macro element in the Design Tree.



Fig. 2.15.16.

The way of the macro element display in the Tree does not affect the display of its member objects in the Model window.

By default the display of a newly created macro element contents is enabled.

To disable the display of a macro element contents, select it in the Design Tree and call from the context menu **Hide Contents** command. To restore the contents display, call the **Show Contents** command from the macro element context menu. If several macro elements with different display settings are selected, then both commands are available in the context menu.

2.15.8.3. Altering the Macro Element Contents

As soon as the macro element has been created, you can change its contents by “dragging” objects in the Design Tree with the mouse.

- ▼ To add an object to a macro element, “drag” the object icon on the macro element icon.



When the cursor shape changes, “release” the object. It will be added to the contents of the selected macro element. If the object was located above the macro element in the Design Tree, the macro element position will not be changed after adding of that object. Otherwise, the macro element is moved to the position of the added object in the Design Tree.

- ▼ To exclude the object from a macro element, expand its contents (to do this, it is necessary to enable the macro element contents display, see Section 2.15.8.2 on p. 840). “Drag” the object icon in the Tree outside the Macro element.



When the cursor shape changes, “release” the object. It shall be excluded from the macro element to pop up in that place of Design Tree, where it had been before inclusion into the macro element.

-
- ▼ To move an object from the macro element into a specific position in the Design Tree, “drag” the object to that position. When the cursor takes on the shape of an arrow, “release” the object. If the cursor doesn’t turn into the arrow, that means the object being moved cannot be put in the specified position in the model object hierarchy.



The central point of the coordinates cannot be included into the macro element nor excluded from it with the mouse. To do this, it is necessary to use the **Group into Macro Element** and **Explode Macro Element**, commands accordingly.

You can also move objects between the macro elements with the mouse.

2.15.8.4. Destruction of the Macro Element

The macro element destruction is the reverse to its creation operation.

After the macro element is destroyed, its icon is deleted from the Design Tree, and its member elements are returned to their places.



The macro elements included into the destroyed one are preserved.

To destroy the macro element (macro elements), select it and call the **Destroy Macro Element** command from the contextual menu or from the **Service** menu.

2.15.8.5. Deleting Macro Element

The macro element deletion means both the element itself and its member objects deletion.

To delete a macro element (macro elements), select it and call the **Delete** command from the context menu or from the **Edit** menu.

After that, the object deleting dialog will be displayed. It lists all the objects included into the macro element. You can confirm or reject the deletion.

2.15.9. Deletion of the model construction history

If necessary, you can delete information about the methods of creating objects and links between them from the model's file. As a result, the model's appearance and its mass-centering characteristics will not change, but it will become impossible to trace back how the model was created and edit parameters of its objects.

Construction history can be deleted if it is no longer needed; for example, if model designing is finished. After deletion of construction history, rebuilding of the model will take less time, and the size of the model's file will be reduced.

The **Delete construction history** can be found in the **Editor** menu. Once the command is called, information about construction of the model will be deleted; the Design Tree will be modified, in particular:

- ▼ bodies/surfaces/curves are transformed into operations/surfaces/curves without history;
- ▼ the array transforms into a set of objects located in accordance with the arrays parameters;
- ▼ sketches, auxiliary axes and planes, layout elements, component matings are deleted;
- ▼ assembly components are fixed;
- ▼ if a component was edited by an operation in assembly, all of its objects are transferred to the assembly, while the component is removed from it;
- ▼ local coordinate systems, control and connection points are replaced by ordinary points.

After history deletion, the model can be edited in a standard way: existing objects can be deleted or edited (using available methods) and new objects can be created. If necessary, history of a construction can be deleted again.



You can cancel deletion of the construction history before you close the model's file.



Deletion of construction history is not available in models with variants. If necessary, you can save variants as individual models (see Section 2.13.3.7 on p. 771) and delete their construction histories.

2.15.10. Creation of Current Model Drawing

You can create drawing with an associative view of model (part or assembly) directly under the work with that model.



Model document should be saved into file on disk.



To create a drawing, call the **Operations — Create New Drawing from Model** command. After that a new drawing is automatically created, and the Default associated view in it. In this case, the current three-dimensional model is already selected as the model displayed in the view. Other view parameters are set by default (orientation — front, color — black, scale — 1:1, etc.). Modify the parameters if necessary.

Then select the position of the base point of view.

An arbitrary view will be created in the drawing depicting the current model.

The further work on the layout, i.e., creating other association views, formatting etc. is performed the same way as with a layout created in a regular manner.

2.15.11. Specifications

Specifications are a text included in the design document. Specifications compliment graphical information and ensure that the respective item is manufactured in strict compliance with the design idea.

In the model document, specifications are prescribed and displayed in a special window (see Section 2.15.11.2 on p. 843). You can transfer them in a model's associative drawing or save in a separate document, as well as print them.

After transferring to the drawing, specifications become part of it and can be edited or moved as needed (see Chapter 3.5.7 on p. 1198). At the same time, link is maintained between specifications in model and in drawing: if needed, changes in model specifications can be transferred to a drawing (see Section 3.6.3.9 on p. 1234).

2.15.11.1. Entering and deletion of specifications in a model

In order to enter specifications, call the **Specifications** command from the **View** menu or from the context menu of the Model Window. The **Specifications** window will appear on the screen. Enter text in the window (see Section 2.15.11.2 on p. 843).

In order to remove specifications from a model, delete the respective text in the **Specifications** window.

Specifications of components are not displayed in the assembly that contains them. You can view or modify specifications of a component in its edit mode **in window** or **at place**.

2.15.11.2. The Specifications window

You can use the **Specifications** window to work with specifications in a model.

This window is enabled or disabled using the **View – Specifications** command.

This window displays specifications if they are assigned to the model. The upper section of the window contains the menu of commands.

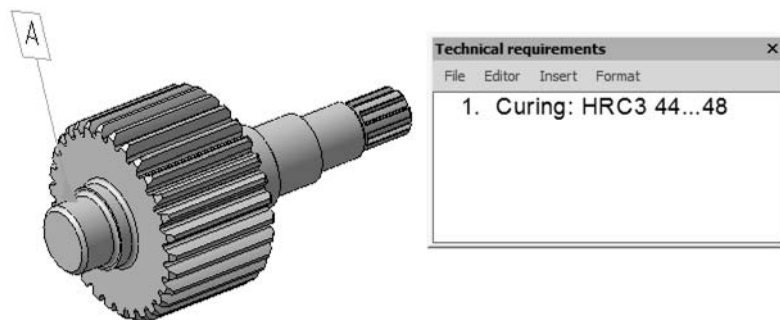


Fig. 2.15.17. **Specifications** window

You can perform the following actions in the **Specifications** window:

- ▼ enter and edit text,
- ▼ insert special objects and format text using commands from the **Insertion** and **Format** menus,
- ▼ format text using commands from the **Format** menu,
- ▼ save text in a separate text file (*.txt) or KOMPAS-3D text document (*.kdw) using the **Save in Document** command of the **File** menu,

- ▼ print text using the **Print** command from the **File** menu.

You can also edit text in the *mode of specifications handling* window. For this purpose, call the **Editor — Edit in window** command from the **Specifications** window menu. A window will appear on the screen displaying model's specifications in the edit mode.

This mode provides the same controls that are available when working with specifications entered in a drawing (see Section 3.5.7.1.1 on p. 1199). You can edit text, paste text fragments, check spelling in different languages and so on.

In the mode of *specifications handling*, text looks the same as it is when printed. When printed, line length depends on paper width. For more information about specifications printing, see Section 2.15.11.3 on p. 844.

Default text parameters are set up for new documents (see Section 9.2.7.9 on p. 1865). In the current document, settings can be adjusted when working with specifications. Text parameters set for model are not transferred to drawing.

In order to exit the *mode of specifications handling*, use special save and close commands in the **File** menu (see Table 3.5.11 on p. 1199).

To close the mode's window and return to the Model Window, you can also click the **Close** system button in the upper right corner of the window.

2.15.11.3. Printing specifications from model

Specifications in model are printed on sheets without layout. They can be printed using one of the following methods.



- ▼ The **Preview** command is used to print specifications together with the model's image. This command can be used for adjusting parameters in the preview mode and printing the document (see Chapter 8.2.1 on p. 1639). In the preview, specifications are displayed in separate sheets. You can adjust scale and other print parameters.
- ▼ When printing specifications without model, you can use the **Print** command which is called from the **Specifications** window or from the *mode of specifications handling* window (see Section 2.15.11.2 on p. 843).

2.15.12. Password for model access

You can protect access to model with a password. Since then, viewing and editing the model will only be possible as long as correct password is entered.

Assemblies (*.a3d) can be protected by entering password for assembly loading type(s). For more information about assembly loading, see Section 2.12.2.2 on p. 735; details about loading type passwords can be found in Section 2.12.2.3 on p. 741.

A **part** (*.m3d) model can be protected by entering password for access to the model. For this, do the following:

1. Highlight the root element (i.e. the part) in the Design Tree.
2. Call the context menu command **Access**.

3. In the appearing **Access to model** dialog, enter the password and a commentary (optional). Close the dialog by click the **OK** button.
4. In the next dialog, enter the password again and close the dialog by clicking the **OK** button.
5. Save the model file.

Password for access to a part, if there is any, will be requested in the following cases:

- ▼ opening of the part's file;
- ▼ insertion of the part into an assembly as a component;
- ▼ attempt to edit:
 - ▼ part — assembly component (using **Edit in window** and **Edit in place** commands);
 - ▼ base part (using the **Edit source** command);
 - ▼ part — image source in the associative view (using the **Edit in window** command).

In all of the above-mentioned cases, the dialog shown in Fig. 2.15.18 will appear. In order to load the model, password has to be entered in the respective field. If necessary, you can choose an access variant: edit or read only.

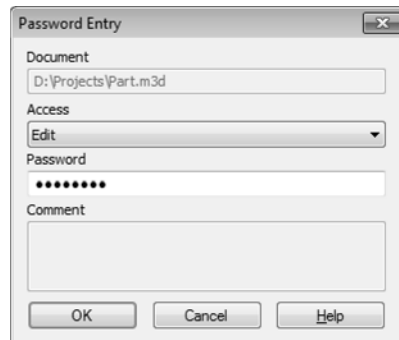


Fig. 2.15.18. Password entry dialog for access to a part's model

Changing and removing password procedures are similar to the password protection procedure. In order to change password, enter the new password in the **Access to model** dialog. If you want to remove password, delete all password symbols. Save the model's file after changing or removing password.

2.15.13. Layers in a model

A layer is a logical group of document's objects. Splitting a model into layers makes it simpler to change the properties of object groups. For example, you can show or hide all objects within a layer at once or transmit them into an associative view, etc.

The system layer is always present in models and, if no more layers had been created by the user, all objects are placed in the system layer.

You can create up to 2,147,483,647 layers in a model. Objects to be placed in a certain layer are selected by the user. Layers can contain objects of any type, including objects from different variants. However, one object cannot belong to two different layers.

To find out a layer where a certain object resides, select the object in the Design Tree or in the Model Window and choose **Object layer** from the shortcut menu. The layers of the selected object will be marked with a tick in the list of layers (see Fig. 2.15.22).

2.15.13.1. Layer statuses

The state of a layer is defined by the values of the following parameter values:

- ▼ This property *Visibility* controls displaying of a view and has two values: **visible** and **hidden**. If a layer is visible, its objects are displayed on the screen.
- ▼ *Color* — this property sets the color of the layer's objects on the screen. The layer color can be assigned in the standard color picker dialog.
- ▼ *Editability* — this property allows or prohibits the editing of the layer's objects and supports two values: **allowed** and **prohibited**.
- ▼ *Projectability into Drawing* — this property allows or prohibits projecting the layer and its objects into associative drawings and supports two values: **allowed** and **prohibited**.



To enable inheriting of the layer property value by the object property, the object property should be set to **By Layer**. For example, suppose that an object is located in *Layer 1*. To hide this object when *Layer 1* is also hidden and display it when *Layer 1* is also displayed, you should set the *Visibility* property of the object to **By Layer**. You can manage the object properties in the **Properties** window (see Section 2.14.3.6.1 on p. 793).

Only one of the layers has the **active (current)** status at a certain moment. The active layers is used for saving new objects.

Any layer can be made current. In this case, it automatically becomes visible. Active layers cannot be hidden until they have this status.

Layer's status and condition does not affect editing of objects. If an object's layer is hidden, the object should be highlighted in the Design Tree. Object editing does not result in any change of the respective layer's condition and status.

If necessary, model's objects can be moved from their layers to any other layer (see Section 2.15.13.6 on p. 850), including hidden ones.



The availability and output of the **Hide/Show** command for an object does not depend on the state of the layer where this object resides. However, note that when the *Visibility* property of an object was set to **By Layer**, then once you call the **Hide** command, it will be set to **Hidden** (and once you call the **Show** command, it will be set to **Visible**).

2.15.13.2. Layers in the Model Document Manager

The Document Manager is used for working with layers in a model. You can use it to perform the following actions:

- ▼ create and delete layers,
- ▼ select current layer,
- ▼ edit property values.

In order to open the Document Manager, perform one of the following actions:



- ▼ call the **Service — Document manager** command,

- ▼ click the **Document manager** button on the **Standard** panel.



- ▼ click the **Layer management** button in the **Current state** bar.

The **Document Manager** dialog for handling layers will appear (Fig. 2.15.19).

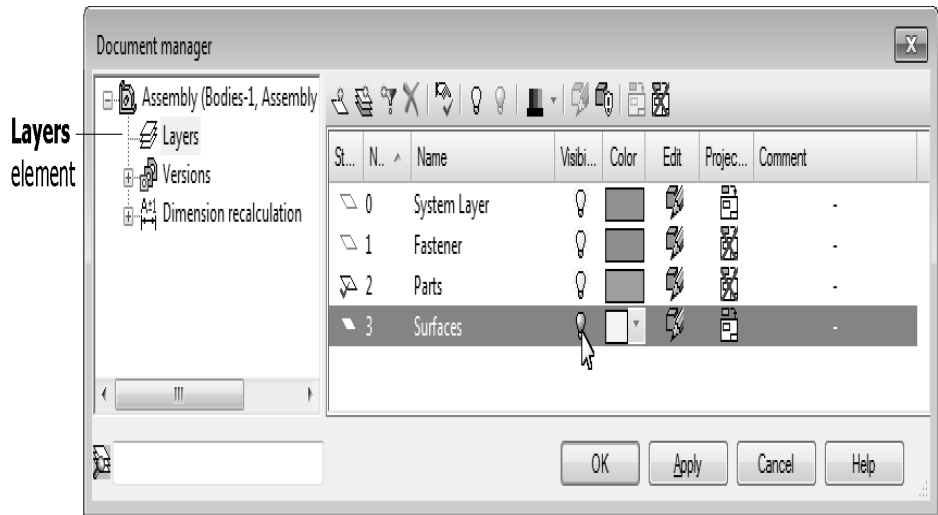


Fig. 2.15.19. Managing layers in the Model Document Manager



The first part of the dialog displays the document's Element Tree. If the dialog is called by the above-mentioned commands, the **Layers** element will be active in the Tree. At the same time, the right section of the dialog will display the list of layers created in the document and a toolbar for handling them.

The list of layers contains the names of layers created in the current model.

Values of the visibility property are displayed as icons in the fields of the List of Layers:

- ▼ *Status*;

- ▼ *Visibility*:



- ▼ visible layer,



- ▼ hidden layer;

- ▼ *color* — as a rectangle with the currently used color;

- ▼ *editability*:



- ▼ allow editing;



- ▼ prohibit editing;

- ▼ *project into drawing*:



- ▼ project into drawing,



- ▼ do not project into drawing.

The current layer is marked with a red check mark in the **Status** field.

An icon of an empty layer has different color than that of the layer with objects: it is gray in the former and blue in the latter.

Clicking a value in the **State**, **Visibility**, **Editing** and **Projection to drawing** field inverses it. Clicking the **Name** field allows modifying the value of the field. The object name can be changed by selecting **Rename** in its context menu. By clicking in the **Color** field, you can assign a color to the layer using the standard color picker.



You can alter the states of several layers at the same time in a similar manner. To do this, select them and then click the necessary button on the toolbar of the Document Manager or call the command from the contextual menu.










You cannot change `and editability` properties.






Objects, located in the List, can be sorted by any of the columns. To select a column for sorting, click on its name. A triangle will appear next to the name. It shows the direction of sorting. If the vertex of the triangle is faced down, then the sorting of the list is done by descending the values in this column, and if up— by ascending.

The toolbar is located in the upper part of the **Document manager** window. Its buttons are described in Table 2.15.4.

Tab. 2.15.4. Document Manager toolbar

The	Function
	Create Layer Create new layer.
	Create Group Create a group of layers.
	Create Filter Create new layer filter.
	Delete Delete selected layers. Deleting a layer means deletion of all objects located on it. Therefore, if you attempt to delete a non-empty layer, the object deletion dialog will appear on the screen. You can confirm deleting the layer with all its objects or cancel the deletion.
	Make current Makes the selected layer active.
	Visible Makes the selected layers visible.
	Hidden Hides the selected layers, i.e. makes them invisible.

Tab. 2.15.4. Document Manager toolbar

	The	Function
	Color	Change Color of Selected Layers. The layer color can be assigned in the standard color picker dialog.
	Enable Editing	Makes the selected layers visible.
	Disable Editing	Makes the selected layers visible.
	Transfer to the drawing	Make the selected layers projectable into an associative drawing.
	Do Not Project into Drawing	Make the selected layers non-projectable into an associative drawing.



The **Search Area** field is located under the Element Tree. This field makes it possible to search a layer by name or number.

In order to begin the search, enter a text line into the field. In this case, you can use masks. The “*” character used in masks replaces any arbitrary number of any characters. “?” character replaces any single character. The layers with names and numbers matching the specified line will be displayed in the List of Layers. The contents of the search area are not saved in the document.

2.15.13.3. Creating a layer in model

In order to create a new layer in model, perform the following operations.



1. Open the Document Manager using one of the above-mentioned methods (see Section 2.15.13.2 on p. 846). For example, lick the **Layer management** button in the **Current state** bar.



2. Click the **Create layer** button on the toolbar of the **Document Manager** dialog.

The new layer will appear in the List of layers. The text cursor will appear in the group name field. You can change the default name and add a commentary to the layer.

By default, a new layer is assigned the first unused number.

3. Set the state of the layer:
 - ▼ *visibility*,
 - ▼ *color*,
 - ▼ *editability*,
 - ▼ *projectability into associative drawing*.

Note that after creating a new layer, the current layer still remains the active one. If you need the new layer to be current, click the **Make Current** button or click the mouse on the **Status** cell. On the layer icon, a red “check mark” will appear, indicating that the layer is current.

4. After finishing entering all new layer parameters, click the **OK** button of the Document Manager.

2.15.13.4. Switching between layers

The current layer's name is displayed in the **Layer Management** field of the **Current State** bar.



The **Layer Management** field is displayed on the Bar, if it is positioned horizontally.

You can make any layer active, or make other layers visible/invisible using the **Document manager** (see Section 2.15.13.2 on p. 846).

However, the following method is the most convenient. Choose or enter using the keyboard the name of the required layer in the **Layer Management** field on the **Current State** bar, and the layer will become active (Fig. 2.15.20).

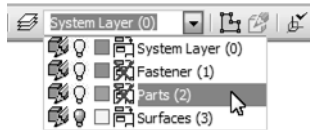


Fig. 2.15.20. Switching between layers on the **Current Status** bar

Objects of the layer selected in the list are highlighted in the Design Tree and in the Model Window. Icons in front of layers' names denote their visibility: *editability*, *visibility*, *color* and *projectability into associative drawing*. You can change the icon's status to the opposite one by clicking it. However, note that for the current layer, you can change only the *projectability into associative drawing* property.

2.15.13.5. Creating a layer in model

To select objects that reside in one or more layers within a model, choose **Select – By Layer**. The submenus with **By clicking** and **From list...** options will appear. These commands are similar to those in the graphical document, see Table 3.5.10 on p. 1192.

2.15.13.6. Moving model's objects between layers

In order to move objects to a certain layer, you can use the **Service – Modify Layer** command. The command can also be called from the context menu.

Once the command is called, control elements for setting layer modification parameters appear in the Property Bar.

Specify the objects that need to be moved to another layer. You can specify components, bodies, surfaces, curves, points, dimensions, designations, sketches, local coordinate systems, and others. Exceptions are axes and planes of the absolute coordinate system which can only be positioned in the system layer.



Operations are not objects that can be moved to layers separately from bodies containing them.

Selected objects are highlighted in the model. Names of objects and their layers are displayed in the **List of Objects** bar (Fig. 2.15.21).



In order to delete an object from the list, highlight the row and click the **Delete** button above the list. To cancel object selection in the model, re-select it.



Objects can be selected before calling the command.

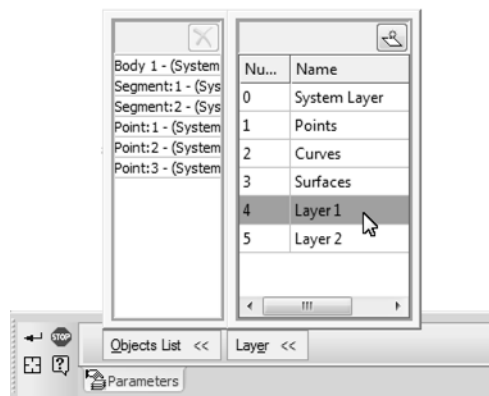


Fig. 2.15.21. Transmitting Objects to Another Layer

In the **Layer** bar, specify the layer to which selected objects have to be moved. For this purpose, click the row with the layer's name.

In order to create a new layer, click the **Create new layer** button above the list.

Once the objects are specified, click the **Create object** button in order to move them to the selected layer.

You can also change a single object's layer by doing the following. Call the **Object Layer** command from its context menu. The command menu contains the system layer and user layers, while the current object's layer is marked with a tick. Specify the required layer in the menu in order to change it (Fig. 2.15.22).



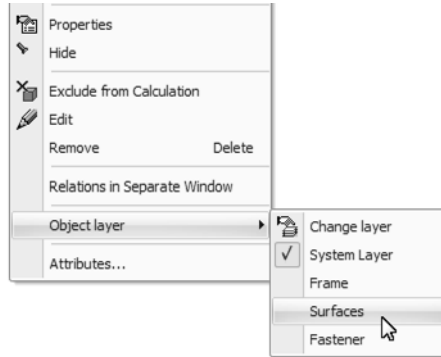


Fig. 2.15.22. Choosing a layer from the **Object Layer** command menu



The value of the property of the object within a layer will be the same as the value of the layer's property, if the object's property is set to **By Layer** in the **Properties** window (see Section 1.4.2.2 on p. 82).

2.15.13.7. Removing a layer from a model

You can delete a layer from the document. Objects positioned on the layer will also be deleted. In order to delete a layer, do the following.



1. Open the Document Manager using one of the above-mentioned methods (see Section 2.15.13.2 on p. 846). For example, lick the **Layer management** button in the **Current state** bar.



2. Select the layers that should be deleted in the List of Layers.

If you know the name of the layer, it can be selected using the Search area located in the lower left corner of the Document manager.



3. Click the **Delete** button on the Document Manager toolbar.



You can delete neither the current nor a system layer.

If the layer being removed contains objects, a dialog will appear on the screen making it possible to view them and confirm deletion, or cancel it.

4. Confirm the deletion.
5. Close the dialog Document Manager by clicking the **OK** button.



When you delete a layer projected into an associative drawing in a document, the layer is also removed from the drawing. If the layer you are deleting contains any non-projective objects, then they are automatically copied to the system layer of the drawing.

2.15.13.8. Layer Groups and Filters in a Model

The purpose of layer groups and filters as well as their workflows are almost the same as in the graphical document. This section briefly describes how to create and remove layer groups and filters in a model. For more details, refer to Section 3.5.6.9 on p. 1192 which describes the sets of layers in a graphical document (in a graphical document, sets of layers include the following: layer groups, layer properties groups and layer filters; models do not support layer properties groups).



To create a layer group in a model, click the **Create Group** button on the toolbar in **Document Manager** (see Fig. 2.15.19). An icon for a new group will appear in the Design Tree.

Drag the layers you want to the icon of the created group.



To create a layer filter in a document, click the **Create Filter** button on the toolbar. After that, the **Layers Filter** dialog will be displayed (Fig. 2.15.23). In the **Conditions** table of this dialog, specify one or more filtering conditions for layers. You can check the results in the **Preview** table.

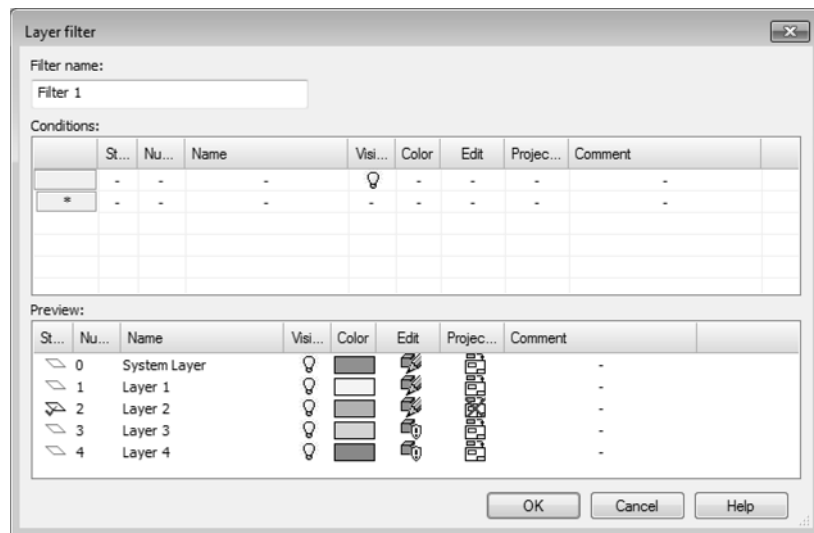


Fig. 2.15.23. The create layer filter dialog for the document model

To finish configuring a filter and close the dialog, click **OK**. The filter icon will appear in the Tree of the Document Manager.



To remove a layer group or a filter, select them in the Tree in Document Manager and click **Delete** or choose the corresponding command from the shortcut menu.

Filters can be converted into groups (see Section *Converting a filter into a group* on p. 1197).

2.15.14. Zones in a Model

You can create zones in a model. Zone is a selected space within the volume of the model.

Zones are useful in models which contain lots of components split into relatively small groups. Using zones, you will be able to select object groups in such models more quickly. For details

on selecting objects using zones, see Section 2.15.14.4 on p. 858.

You can split a zone into multiple zones. The resulting zones, in turn, can also be split into zones, and so on.



You can display or hide zones in the model window by choosing **View – Display – Show Zones** or by clicking the **Display Zones** on the **View** toolbar.

All zones available in the model are listed on the **Zones** tab in the Model Tree (see Section 2.15.14.1 on p. 854). When this tab is active, the zones are displayed in the model window independently of their visibility settings.

Zones are shown in the model as semi-transparent polyhedrons. You can set the zone visibility parameters for the current window using the special dialog (see Section 9.3.4 on p. 1867).

When you create zones in the variant models, they are transmitted just like any other objects (see Section 2.13.3.2 on p. 761).

2.15.14.1. Zone Tree

The zone tree is displayed on the **Zones** tab of the Model Tree (Fig. 2.15.24). The tree displays the zones and splittings starting from the oldest ones.

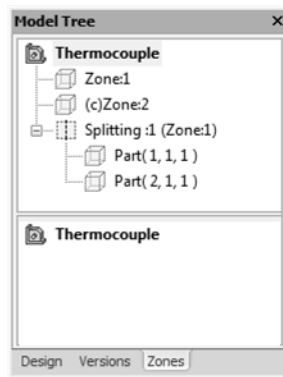


Fig. 2.15.24. Model Zones Tree

Zones and splittings are shown at the first level of hierarchy in the Tree.

The subzones derived from splittings the zones are shown at the second level.

The zones and splittings are automatically assigned the names and numbers during creation.

Letter 's' in parenthesis precedes each system recalculation. To make another zone the current one, select it in the Zone Tree and choose **Current Zone** from its shortcut menu. Upon calling the command, a “tick” appears beside its name in the menu.

To hide the assembly components, highlight them in the model's Tree and call the **Hide** or **Show** command in the contextual menu.

To remove a zone or a splitting, select the required object in the Zone Tree and choose **Delete** from the shortcut menu.

2.15.14.2. Creating a Zone in a Model


A zone is an object that uses coordinate system (for object coordinate system, refer to Section 2.8.3.4.1 on p. 598).

You can set the zone dimensions in a model in the following ways:

- ▼ **By coordinates;**
- ▼ **By object dimensions.**

For your convenience, you can display the zones in the model window or choose the **Zones** tab in the Model Tree.

Call the **Create Zone** command in one of the following ways:

- ▼ from the **Editor — Zones** menu,
-  ▼ by clicking the corresponding button on the **Construction Geometry** toolbar,
- ▼ from the shortcut menu of a part or an assembly on the **Zones** tab.

Once you call the command, the **Parameters** tab on the Properties panel will display the controls for adjusting the zone parameters, see Fig. 2.15.25.

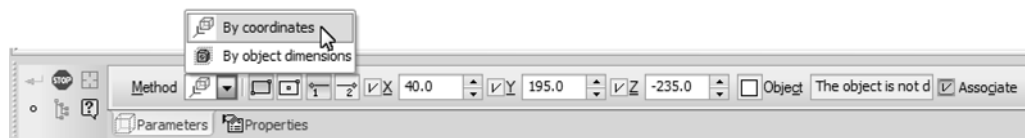


Fig. 2.15.25. Properties panel during creation of a zone

From the **Method** drop-down list, choose how to set the zone dimensions and specify the required parameters. Parameter input for each method is detailed in Sections 2.15.14.2.1 on p. 855 — 2.15.14.2.2 on p. 856.

You can specify the objects for setting the zone dimensions in the Design Tree or in the Model Window before calling the command. In this case, the Properties panel will display the controls for **By object dimensions** by default.

On the **Properties** tab of the Properties panel, you can change the default name of any zone. This tab also displays the name of the curve coordinate system.



This tab also displays the name of the zone coordinate system.

To set other parameters of the zone without leaving the command, click **Select other** on the Advanced Control Panel.



To finish building, click **Create object** button.

The created zone will appear in the model window while its icon and name will appear in the Model Zone Tree.

2.15.14.2.1. Creating a Zone by Coordinates



To set the zone dimensions using the **By coordinates** method, perform the following actions:



1. Choose the box definition type. To do this, enable one of the following switches in the **Option** group:
 - ▼ **By two vertices,**



▼ By center and vertex,

2. Position parallelepiped points.



The position of the first point is set when the **Point 1** switch is active and that of the second one — when the **Point 2** switch is active.



Point coordinates can be input from a keyboard or defined automatically by positioning the point with the mouse in the model window or selecting a point object. These operations are similar to those described in Section 2.5.2.2 on p. 336.



You can also specify the coordinates using the point construction process which can be launched by clicking the **Point Creation** button on the Advanced Control Panel (see Section 2.5.2.11.1 on p. 347). The parameters set during this process define the position of the box point. No new point is created in the process.

2.15.14.2.2. Creating a Zone by Object Dimensions



To set the zone dimensions using the **By object dimensions** method, specify the objects which define the zone dimensions.

The objects can be selected either in the Model window or in the Design Tree. The specified objects are highlighted, and their names appear in the **List of objects Bar**.

The auxiliary planes and axes are not used for defining the zone dimensions.



To delete any object from the list, highlight it and press the **Delete** button, located above the list.

Second selection of the base surface cancels its selection.

The zone looks like a box which covers all the objects you've specified.

When choosing objects in the Design Tree, you can specify either the entire model or any of its components. In this case, the default zone dimensions are defined by the bodies and surfaces included in the model or its component. During the configuration process, you can change the list of objects which define the zone dimensions (see Section 9.2.7.4.14 on p. 1858).

The lists of object types for the model and its components may be different. In this case, the zone dimensions for a model are defined only by the objects included into all these lists. If a zone dimension does not take into account some types of objects when you select the component, this means that they were excluded from the list during configuration process for this component. You can set the required components manually when required.



When the **Specify Parts** button is pressed on the Advanced Control Panel, you can select individual parts at any assembly level. When this button is released, then when you select a part, the entire component of the first level to which this part belongs is selected.

2.15.14.3. Splitting into Zones

You can split a zone into subzones in the following ways:

▼ Uniformly along axes;

▼ By set of planes.

For your convenience, you can display the zones in the model window or choose the **Zones** tab in the Model Tree.



To split a zone, choose **Editor — Zones — Split Zone** or click the corresponding button on the **Construction Geometry** toolbar.

Once the command is called, control elements for setting the zone splitting parameters appear in the Properties panel (see Fig. 2.15.26).

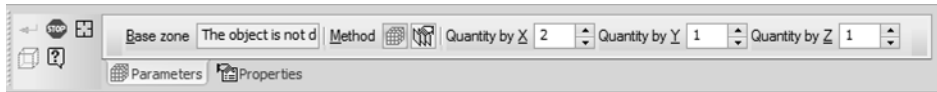


Fig. 2.15.26. Properties panel when splitting a zone

Specify the sketch in the Model Tree or Window.

On the Properties panel, select the zone splitting method and set the required parameters in the **Splitting Method** group of switches.

Parameter input and object specification for each method is detailed in Sections 2.15.14.3.1 on p. 857— 2.15.14.3.2 on p. 857.

Once you've specified the parameters, the model window will display the ghosts of the new zones in the zone being split.

On the **Properties** tab, you can change the default name of the splitting.



To set other splitting parameters of the zone without leaving the command, click **Select other** on the Advanced Control Panel.



You can create a new zone in the model without leaving the command. To do this, click the **Create zone** button on the Advanced Control Panel.



To finish building, click the **Create object** button.

The new zones will be displayed inside the zone being split.

The names of the splitting and new zones are shown in the Model Zone Tree.

A single zone can contain multiple splittings.

2.15.14.3.1. Splitting a Zone by Axes Equally



To split a zone using the **Equally by axes**, specify the values you want in the **Number of Zones by X**, **Number of Zones by Y** and **Number of Zones by Z**.

The number of zones on at least one of axes must be greater than 1.

2.15.14.3.2. Splitting a Zone by A Set of Planes



To split a zone using the **By set of planes**, specify the planes which will divide the zone. The planes must intersect with the zone you want to split. The following can be used as a plane:

- ▼ coordinate and auxiliary planes,
- ▼ flat faces.

The planes can be selected either in the Model window or in the Design Tree. The specified objects are highlighted, and their names appear in the **List of objects**. Bar.



To delete any object from the list, highlight it and press the **Delete** button, located above the list.

Second selection of the base surface cancels its selection.

2.15.14.4. Selecting Objects Using Zones

To select objects using a zone, perform the following actions.

1. Display the zones in the model window or choose the **Zones** tab in the Model Tree.
2. Specify the required object in the Design Tree or Model Window.
3. Choose the required zone from the **Select — By zone** menu or from the context menu:



- ▼ **Select Inside a Zone** — choose the objects within a zone, including the tangential ones.
- ▼ **Select Outside a Zone** — choose the objects outside a zone, including the tangential ones.
- ▼ **Select with a Section Zone** — choose the objects inside the zone as well as objects intersecting the borders of the zone.

If no zone is selected when running the command, then the command is applied to the current zone.



When selecting by zone, the object filters are taken into account (see Section 2.1.4.1.3 on p. 133).

You can use the object group selection by zones to apply the certain commands to all the selected objects at once, e.g. change the loading type (for details on loading types, see Section 2.12.2.1.6 on p. 733).

After running any of the selection command, the list of selected objects is displayed in the lower part of the Zone Tree. When you call a command from the shortcut menu of any object in the list, the command is applied to all objects at once. For example, when you call the **Loading types** command and specify a certain loading type, this type will be applied to the entire object group.

You can use selection by zones either at the level of the current document or at all levels (this can be configured, see Section 9.1.11.14 on p. 1738). In the first case, the system checks whether the components of the model are included in the zone, and in the second case, the system checks object by object. Therefore, in the first case you can choose only the entire components while in the second case you can select individual objects within them.

2.16. Technological preparation of a model

2.16.1. Technological assembly

KOMPAS-3D makes it possible for the user to prepare a model (part or assembly) for the development of the technological process of its manufacturing (or assembly). For this purpose, the technological assembly is used.

The **Technological assembly** is a 3D model that contains technological data, for example, the result of model size recalculation taking account of tolerances, technological objects (center holes, mounting holes, and so on), technological models (lunettes, centers, instruments, and other equipment). The technological assembly is created and saved in a Technological Assembly file with the *t3d* extension.

Examples of using the technological assembly:

Example 1

A part has been created by the designer. In order to develop the technological process of the part's manufacturing, the technologist creates the technological assembly, incorporates the part in it, and adds the required technological data.

Example 2

The general gear box assembly has been created by the designer. Two cases of the gear box should be processed together. Models of the cases are located in different subassemblies of the gear box's general assembly. The technologist creates the technological assembly and incorporates the gear box's cases in it. Then, the technologist adds technological data to the assembly that is required for joint processing of the cases.

The technological assembly may contain any objects that can be part of an ordinary KOMPAS-3D assembly (see Section 2.1.1.1 on p. 99).

Working with technological assembly (adding components, creation and editing of bodies and surfaces, creation of mates and so on) is similar to working with ordinary assembly.



Some toolbars in the technological assembly are disabled by default (for example, the **Arrays** bar).

The following procedure for creating a technological assembly is recommended.

1. Create the new technological assembly document.
2. Add the required components (created previously by the designer of a part and/or assembly).
3. If necessary:
 - ▼ edit the model by performing operations in the technological assembly,
 - ▼ recalculate dimensions of the model based on tolerances;
 - ▼ add:
 - ▼ technological models and standard items in the form of components,
 - ▼ technological data.



If, when working with the technological assembly, the assembly's contents should be changed, it is recommended to fix its components. For this purpose, you can highlight the *Components* section in the Design Tree and click the **Enable fixation** button on the **Edit assembly** bar.



Component changes performed by operations in the technological assembly are not transferred to the component's file. These changes are saved in the technological assembly. At the same time, all changes in the model, made during modifications of the model file, will be transferred to the technological assembly.



If the technological assembly is created on the basis of an ordinary assembly using the **Save as...** command, all other changes of the initial assembly will not be transferred to the technological assembly.

By default, components are added to the technological assembly with nominal dimensions. You can change dimensions of each component in the assembly taking account of tolerances assigned by the designer to the component's dimensions (dimensions of sketches and dimensions corresponding to parameters of operations; see Section 2.9.2.2 on p. 632). More information about managing the dimensions of a component in an assembly can be found in Section 2.16.2 on p. 860.

2.16.2. Taking account of tolerances when controlling dimensions of a component

By default, components are added to the model with nominal dimensions. If necessary, you can recalculate the dimensions of a component taking account of assigned tolerances (see section 2.10.2 on p. 667 for more information about tolerances). Dimensions of a component are calculated in accordance with the current recalculation.

KOMPAS-3D supports three system recalculations of dimensions: **to middle part of the tolerance field**, **by upper limit** and **by lower limit**. The component can also have **custom** recalculations creating in the process of its editing. For more information on recalculations of dimensions, see Section 2.10.3.1 on p. 672.

You can control dimensions of a component taking into account the tolerances after the component has been inserted.



You can adjust recalculation of dimensions only for components of the first level (i.e. for a component inserted directly in the current model). Defined parameters of recalculation of a component's dimensions are applied to dimensions of all objects included in it.

2.16.2.1. Recalculation of a component's dimensions within an assembly

Recalculation of a component's dimensions in an assembly is initiated by the **Component properties** command. The command is available in the context menu of the component selected in the model window or in the Design Tree.

When the command is called, the pages containing control elements for component properties setup will be displayed on the Property bar.



In order to modify the component's dimensions in accordance with the required recalculation, click the **Recalculate Dimensions** button on the **Source File** tab. A list of recalculations will become available (see Section 2.10.3.1 on p. 672). Choose the required recalculation from the list – a system recalculation or a user one (Fig. 2.16.1).

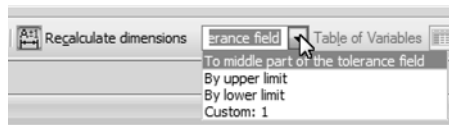


Fig. 2.16.1. Selection of a component's dimensions recalculation



Note that only system recalculations are available for a local part.



Having finished selection of a recalculation, click the **Create object** button on the Advanced Control Panel.

The icon of the edited component image will be marked in the Design Tree by a red “check-mark” which means the assembly needs to be rebuilt.



In order to see how the assembly looks like with modified parameters of the component, click the **Rebuild** button in the **View** bar or call the **View – Rebuild** command.

In the Design Tree, the recalculated component is marked with letter “r” in parentheses.

2.16.2.2. Restoring nominal dimensions of component.

To restore nominal dimensions of the component that was inserted into a model, take the following steps.



1. From the component's context menu, call the **Component properties** command.



2. In the **Source File** tab, disable the **Recalculate Dimensions** button. The list of recalculations will become unavailable.



3. In the Special Control bar, click the **Create Object** button.

4. Rebuild the model. For this purpose, click the **Rebuild** button on the **View** bar or call the **View – Rebuild** command.

3. Drawing. Drawing Layouts

3.1. General working techniques for graphical documents

3.1.1. Snaps

3.1.1.1. General Information on Snaps

While working with a graphic document, you constantly experience the need to place the cursor precisely at a specific point (a coordinate system origin, a circle center, a segment end, etc.), in other words, to snap to existing points or objects. Without such a snap, you cannot create a precise drawing.

KOMPAS-3D offers snaps to **defining points** (intersection, border points, center, etc.) and **objects** (by normal, by coordinate axes directions).

Snaps can be made by means of the keyboard (keyboard snap) or using special commands. Keyboard snap is available any time while command-based snap is only available when creating or editing graphical objects.

Command-based snap may be attributed **globally** (global snap) or **locally** (local snap).

The global snap (if enabled) works always as you create and edit objects. For example, if the global snap to intersections is enabled, then when creating each point, the system will automatically search for the nearest intersection within the reach of the cursor's catcher.

Local snap has a single time effect. For example, if the local snap to intersections is enabled while creating a point, then after completing creation of that point the snap stops working. To make the same snap when creating the next point, it should be re-enabled. This is inconvenient to use local snap if you need to make several similar snaps in a row.



Local snap has higher priority than the global one, i.e. global snaps enabled in the window have no effect when local snap is in effect.

List of keyboard snap shortcuts is presented in Section 3.1.1.3 on p. 870, list of global and local snap commands— in Section 3.1.1.2.

3.1.1.2. Global and Local Snap

List of global and local snap commands used in KOMPAS-3D is shown in Table 3.1.1.

Tab. 3.1.1. Global and Local Snap commands

Name	Description
Nearest Point	Snap to a defining point of an object (e.g., the start point of a line) or the origin of the current coordinate system.
Midpoint	Snap to the midpoint of an object.
Intersection	Snap to the intersection of objects.

Tab. 3.1.1. Global and Local Snap commands

Name	Description
Tangency	When selecting this method, the snap will be performed in such a way that the created object (line segment, arc, etc.) touches the specified object in the point nearest to the current cursor position.
Normal	When selecting this method, the snap will be performed in such a way that the object being created (such as a line segment) will be positioned perpendicular to the specified object.
By Grid	Snap to a point in the construction grid (see Section 3.1.4.1 on p. 885) in the current window (the grid may be enabled or disabled).
Alignment	When you select this snap method, the entered object point will be aligned vertically and horizontally relative to the defining points of the existing objects and the last fixed point. Alignment is done without respect to the slope angle of the local coordinate system.
Angular Snap	When selecting this snap method, the cursor will move at an angle to the last fixed point, which is a multiple of the value defined among the snap settings (see Fig.3.1.3). The angles are measured in the current coordinate system. By default, the angular snap step is equal to 45°.
Centerpoint	Snap to the center point of a circle, ellipse, circular or elliptic arc, rectangle, regular polygon.
Point on Line	Snap to the nearest point on the selected curve. The nearest point will be determined as an intersection of the curve with a normal drawn from the specified point.

The snap is done as follows.

During the graphical object creating and editing set the cursor so that its catcher reaches to the object (or point), which you need to snap to (Fig.3.1.1). For example, to use **Nearest Point** snap, you have to capture a defining point of any object (a few examples of defining points are shown on Fig.3.4.2 on p. 1115); to use **Center** snap— an arc, polygon, or different object with a center point.

A “cross” will appear at the point corresponding to the snap, which means the snap has been activated (Fig.3.1.1).



When using global snaps, the only one snap with the highest priority will be enabled (see Section 3.1.1.2.1).

To finish the snap press the <Enter> key or the left mouse button. The point marked by a “cross” will be fixed.

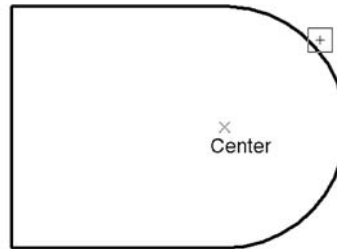


Fig. 3.1.1. Using the **Center** snap to specify the hole center



You can change the size of the cursor's catcher by calling the **Service — Parameters... — System — Graphic Editor — Cursor** command (see Section 9.1.7.1 on p. 1703).

3.1.1.2.1. Using Global Snaps



To manage global snaps, use the **Snaps** button menu in the **Current Status** tab. To enable a desired snap in the current window, call the relevant menu command.

To enable snaps you can also use **Global Snaps** bar buttons. While the button is in depressed state, the snap will be working. To disable a snap, release the button.



Fig. 3.1.2. Global snaps bar

You can turn on several different global snaps to objects, and all of them will work simultaneously. In this case, the point is calculated “on-the-fly”, and a ghost is displayed that corresponds to this point.

If several snaps can be enabled in the current cursor position, the one with the highest priority will be enabled. The list of priorities coincides with the order of listing the snaps in their setup dialog (Fig. 3.1.3).

Suppose that the snaps **Nearest Point** and **Intersection**, which follow one after another in the list, are activated. If in the current cursor position (for example, when selecting a point for alignment) its “catcher” reaches to a defining point of an object and an intersection point of objects, then the higher-priority **Nearest Point** snap will become activated.

The snaps **Orthogonal**, **Align** and **Point on Curve** (if enabled) can be activated in couples. For example, the combined use of the **Align** and **Point on Curve** snaps allows fixing a point on a curve with the same X or Y axis as the defining point of any object.



You can disable all global snaps and then re-enable them by clicking the **Snaps** button. The Disable Snaps button serves as an indicator of enabled global snaps: the pressed button means the global snaps are enabled, the released button means they are disabled. To switch the button from the keyboard, use the key combination **<Ctrl>+<D>**. To temporarily disable global snaps, press and hold the **<Alt>** key; once the key has been released, the snaps will be enabled.

Enabling and disabling of the global snaps as well as a number of additional settings are available in the global snap setup dialog (Fig.3.1.3). You can open it by clicking the **Set Global Snaps** button on the Current Status bar.

The control elements of this dialog are described in Section 9.1.7.16 on p. 1717.



Fig. 3.1.3. Global snap setup dialog

Note that enabling and disabling of the global snaps using described above methods will be active only in the current window till the end of the current working session.



If a document is open in several windows, then the snap settings made in one of them affect all other windows of this document.

The default setting of global snaps — the one used in the windows of newly created or open documents — will remain unchanged. In order to change it, call the **Service — Parameters...** command. — **System— Graphic Editor— Snaps** command. A dialog shall appear, with controls identical to those shown in Fig. 3.1.3 (the only exception is the absence of the **Disable Snaps** option), where you can enter the required settings.

3.1.1.2.2. Local Snap use

A local snap enabling is possible while you create and edit graphic objects. The commands for enabling local snaps are grouped together in the menu. There are two ways to access this menu.

1. Using the context menu (Fig. 3.1.4).

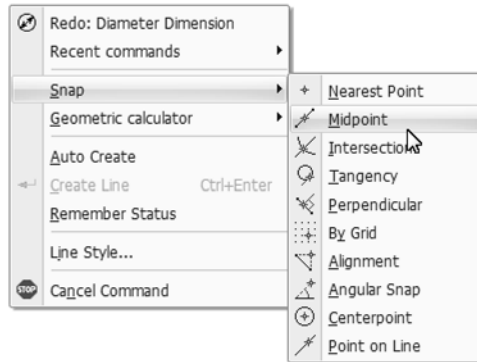
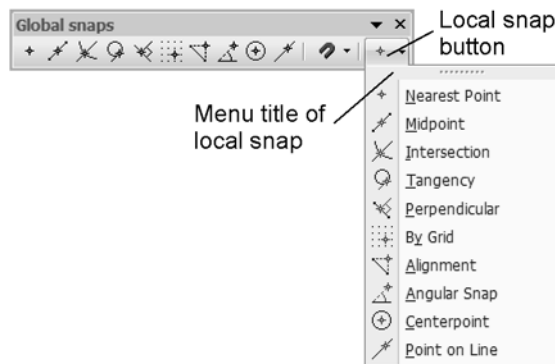


Fig. 3.1.4. Commands for enabling local snaps in the context menu

- Using the **Local Snaps** button. This is the button of the last used local snap on the **Global Snaps** bar (Fig. 3.1.5).

Fig. 3.1.5. Menu of the **Local Snaps** button

The commands in the **Local snaps** button's menu can be placed as buttons on a separate bar, and then move this bar to any convenient location. To do this, drag the **Local Snaps** menu button by its title in any direction.

The **Local Snaps** toolbar will be created. Note the difference of the Orientation bar from other toolbars: you cannot change the composition or the order of buttons on them.

To enable local snap needed call the corresponding menu command or press the button on the **Local Snaps** bar.



Cursor will change its shape, indicating that the snap is active. All global snaps will be suspended until the current operation is finished.



The **Local snaps** button on the **Global snaps** bar is available after calling the object creation or editing command.

3.1.1.3. Keyboard Snaps

Some snaps can be activated using the keyboard by pressing the corresponding keyboard shortcuts. These commands are shown in the Table 3.1.2.

Keys <0>–<9> and <.> have to be pressed on the additional keypad. The keypad has to be in numeric mode (the NumLock indicator has to be on).

Tab. 3.1.2. Snap shortcuts

Combination	Description
<Ctrl>+<0>	Move the cursor to the (0,0) point of the current coordinate system.
<Ctrl>+<5>	Position the cursor in the element defining point nearest to it, independently on what is on Background Views and Layers*
<Ctrl>+<Shift>+<5>	Position the cursor in the element defining point nearest to it, including what is on Background Views and Layers.
<Ctrl>+<.>	Position the cursor normal to the nearest point of the nearest element.
<Alt>+<5>	Position the cursor at the nearest to it point of two entities intersection.
<Shift>+<5>	Position the cursor at the nearest to it mid-entity.
<Ctrl>+<1>	Snapping to the nearest element along the diagonal between the negative direction of the OX axis and the negative direction of the OY axis of the current coordinate system.
<Ctrl>+<2>, <Ctrl>+<⤴>	Snapping to the nearest element in the opposite direction of the OY axis of the current coordinate system.
<Ctrl>+<3>	Snapping to the nearest element along the diagonal between the positive direction of the OX axis and the negative direction of the OY axis of the current coordinate system.
<Ctrl>+<4>, <Ctrl>+<⤵>	Snapping to the nearest element along the opposite direction of the OX axis of the current coordinate system.
<Ctrl>+<6>, <Ctrl>+<⤶>	Snapping to nearest element in the positive direction of the OX axis of the current coordinate system.
<Ctrl>+<7>	Snapping to the nearest element along the diagonal between the negative direction of the OX axis and the positive direction of the OY axis of the current coordinate system.
<Ctrl>+<8>, <Ctrl>+<⤷>	Snapping to the nearest element in the positive direction of the OY axis of the current coordinate system.

Tab. 3.1.2. Snap shortcuts

Combination	Description
<Ctrl>+<9>	Snapping to the nearest element along the diagonal between the positive direction of the OX axis and the positive direction of the OY axis of the current coordinate system.

* This key combination is also available when the NumLock mode is disabled. In this case, it serves to position the cursor in the element defining point nearest to it, including what is on Background Views and Layers

3.1.2. Geometric Calculator

One of the ways to specify numerical parameters of graphic objects is to copy the parameters of existing objects. To do this, use the so-called **Geometric Calculator**.

The geometric calculator commands are available in the context menu after you enter a numeric parameter if the **text cursor is not located** in this field. The set of commands depends on the parameter type (see Table). 3.1.3 on p. 874).

After executing the Geometric calculator command, you should specify the object(s), whose parameters you want to obtain: locate the cursor so that it's "trap" captures the required object and press the <Enter> key or left-click the mouse.



In order to change the cursor's "catcher" size, use the **Service — Parameters... — System — Graphic Editor — Cursor** command.



The Properties bar changes while the geometric calculator is in use. All tabs on the Properties bar are replaced with a single bar whose name corresponds to the selected command of the geometric calculator, and the **Interrupt Command** and **Help** buttons remain on the special control panel. They allow canceling the geometric calculator or receiving advice on the current geometric calculator command, respectively.

Let us consider ellipse construction as an application example of the geometric calculator, with ellipse center being at the point of intersection of line segment extensions, it's axis parallel to one of the segments, and it's semi-axis equal to the arc length (Fig. 3.1.6).



Fig. 3.1.6. Objects, the parameters of which shall be used by the Geometric calculator

Create a new graphic document and construct the objects shown in Fig. 3.1.6, trying to visually maintain the proportions. Then, complete the following actions.

1. Click **Tools — Geometry — Ellipses — Ellipse** command.

Fields will appear on the Properties bar for entering the ellipse parameters — center coordinates, semi-axes end coordinates, semi-axes lengths, and the slope angle of the first semi-axis (Fig. 3.1.7).

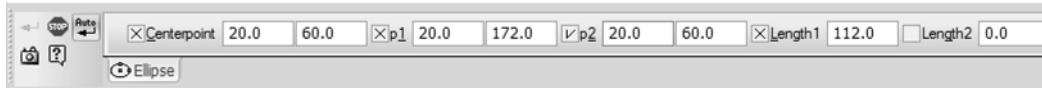


Fig. 3.1.7. Property bar when creating a line segment

2. Open the context menu in the coordinate field of the ellipse center. This menu contains geometric calculator commands for specifying the point location (Fig. 3.1.8).
3. Call **At Intersection of Curve Extensions** command from this menu.

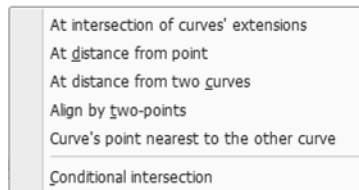


Fig. 3.1.8. Geometric calculator menu when specifying the point

The **Ellipse** tab on the Properties bar shall be replaced with the **At Intersection of Curve Extensions** tab.

4. Select with the cursor the line segments, at the extensions of which the ellipse center should be created. The intersection point of line segment extensions is built automatically.

The **Ellipse** tab will appear on the Properties bar again.

The point coordinate value will be calculated, entered in the **Center** field and fixed.

5. Open the context menu in the field of the slope angle of the first ellipse axis. This menu contains geometric calculator commands for specifying lengths (Fig. 3.1.9).

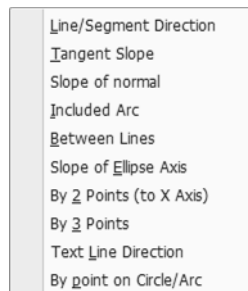


Fig. 3.1.9. Geometric calculator menu when specifying the angle

6. In this menu, call the **Line Direction** command and proceed to specify the line parallel to which the first ellipse axis should be.

The slope angle value of this line relative to the OX axis of the current coordinate system will be calculated, entered in the **Angle** field and fixed.

7. Open the context menu in the field of the first ellipse axis length. This menu contains geometric calculator commands for specifying lengths (Fig.3.1.10).

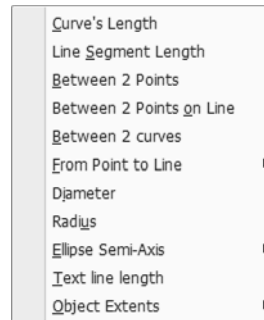


Fig. 3.1.10. Geometric Calculator menu

8. In this menu, call the **Line Length** command and proceed to select the arc to which the ellipse semi-axis length should be equal.

The length value will be calculated, entered in the **Length 1** field and fixed.

9. Specify the length of the second semi-axis on your own.

At this point, the construction of the ellipse with the specified parameters is complete (Fig. 3.1.11).

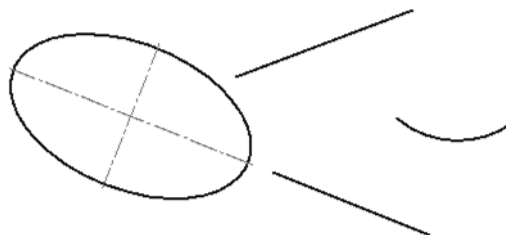


Fig. 3.1.11. Ellipse construction result



If the value was fixed before you executed the geometric calculator command, a new value will be entered in the parameter field by the calculator and the fixation will be preserved.



When specifying coordinates during object creation (for example, when defining the start and end points of a line segment), you can call the Geometric calculator commands for point creation (fig.3.1.8 on p. 872) not only from the context menu for coordinate fields of this point, but also from the context menu of a document window.

Tab. 3.1.3. Geometric calculator commands

Command	Purpose
When entering coordinate values	
At Intersection of Curve Extensions	Obtaining coordinate values of the point located at the intersection of extensions of the specified elements.
At Distance from Point	Obtaining coordinate values of the point located at the preset distance from the reference point.
At Distances from Two Curves	Obtaining coordinate values of the point located at the preset distances from specified elements.
Align by Two Points	Obtaining coordinate values of the point aligned by two points. The point is considered to be aligned by two other points if they are the point's projections on the current coordinate system axis.
Intersection Notation	Obtaining coordinate values of the point located at the intersection of extensions of the specified elements with intersection notation construction. For details about intersection notation see Section 3.3.12.19 on p. 1065.
Curve's Point, Nearest to Another Curve	Snapping and obtaining coordinate values of the point located on the specified element and nearest to another specified element.
When entering linear values	
Line Length	Copying the length (perimeter) value of the selected element.
Line Segment Length	Copying the length value of a polyline or layout segment as well as a polygon side.
Between Two Points	Serves to read the distance between two selected points.
Between Two Points On Line	Copying the length of a curve section limited by two points selected on this curve.
Between Two Lines	Copying the value of the minimum distance between two selected objects.
From Point to a curve — By Normal	Copying the value of the shortest distance between the selected point and the curve.
From Point to a curve — Along Y	Copying the value of the distance between the selected point and object in the positive direction of the Y axis of the current coordinate system.

Tab. 3.1.3. Geometric calculator commands

Command	Purpose
From Point to a curve — Against Y	Copying the value of the distance between the selected point and object in the negative direction of the Y axis of the current coordinate system.
From Point to a curve — Along X	Copying the value of the distance between the selected point and object in the positive direction of the X axis of the current coordinate system.
From Point to a curve — Against X	Copying the value of the distance between the selected point and object in the negative direction of the X axis of the current coordinate system.
Diameter	Copying the diameter value of the selected circle or arc.
Radius	Copying the radius value of the selected circle or arc.
Ellipse Semi-axis— Major	Copying the length of the major semi-axis of the selected ellipse.
Ellipse Semi-axis— Minor	Copying the length of the minor semi-axis of the selected ellipse.
Text Line Length	Copying the length of the selected text line.
Object Extents— Horizontal	Copying the value of the horizontal dimension of the selected object.
Object Extents— Vertical	Copying the value of the vertical dimension of the selected object.
When entering angular values	
Line Direction	Serves to read the angle between an infinite line or segment and the positive direction of the X axis of the current coordinate system.
Tangent Slope	Copying the value of the angle between the tangent to a selected element passing through the selected point on this element and the positive direction of the X axis of the current coordinate system.
Normal Slope	Serves to read the angle between the normal to an element going through a specified point on that element and the positive direction of the X axis of the current coordinate system.
Included Angle	Copying the included angle value of the selected circular arc.
Between Lines	Copying the value of the angle between two selected lines.
Slope of the Ellipse Axis	Serves to read the angle between the major axis of the specified ellipse and the positive direction of the X axis of the current coordinate system.

Tab. 3.1.3. Geometric calculator commands

Command	Purpose
By Two Points (with the X Axis)	Serves to read the angle between the imaginary line going through two specified points and the positive direction of the X axis of the current coordinate system.
By three Points	Copying the value of an angle formed by three selected points. Selection procedure: angle vertex, points on its sides.
Text Line Direction	Serves to read the angle between the line of text and the positive direction of the X axis of the current coordinate system.
By Point on Circle/ Arc	Serves to read the angle between the radius constructed from the circle center to a specified point on the circle and the positive direction of the X axis of the current coordinate system.

3.1.3. Selection of Objects. Using the Clipboard

3.1.3.1. Selecting Objects with the Mouse

To select items by the mouse, complete the following steps.

1. Move the cursor to the required object so that the cursor “trap” will capture the object.
2. Click the left mouse button. The object color will change— it will be highlighted with the color set for the selected objects (see Section 9.1.7.11 on p. 1713).

To de-select an object, click the left mouse button anywhere outside that object or press the *<Esc>* key. The selection will be removed — the object will revert to its original color.

If you need to select several objects, press the key *<Shift>* or *<Ctrl>* and, while holding it down, click the left mouse button on the desired objects. After completing your selection, release the *Shift* key (*Ctrl*).

You can select several objects using another method — by the **surrounding** or **cutting** marquee. Set the cursor at a free space (so that it does not reach to any objects), press the left mouse button and drag the cursor while holding the mouse down. The marquee will be rubber banding on the screen following the cursor.

- ▼ The surrounding marquee is formed when you move the cursor from left to right. It is displayed as a solid line. After you release the mouse button, those objects will be selected that are entirely inside the marquee frame.
- ▼ The cutting marquee is formed when you move the cursor from right to left. It is displayed as a dotted line. After you release the mouse button, those objects will be selected that are entirely or partially inside the marquee frame (the latter means those were intersected by the frame).

To deselect individual objects, click them with the left mouse button, while holding down the *<Shift>* or *<Ctrl>* key. You can deselect multiple objects by a rectangular marquee frame. This is done similarly to selecting, but with the *Shift* key pressed.



When deselecting by a marquee frame, watch that only selected objects are within the frame. Otherwise, the selection is not cleared; rather, it is applied to the previously not selected objects.

Sometimes, the objects that need to be selected are positioned closely to each other or even overlapping each other. At the same time, it is difficult (or sometimes utterly impossible) to precisely select one of them by the cursor.

The command **Object Picking Over** is used for selecting any of the nearby objects (including superimposed objects) (see Section 3.1.3.4 on p. 881).

3.1.3.2. Selecting Objects by Commands

The graphic objects selection commands are grouped together in the **Select** menu, and the commands for removing the selection — in the **Select — Exclude** menu.

The buttons for executing the commands are located on the **Selection** bar (Fig.3.1.12).



Fig. 3.1.12. **Selection** bar

The general selection commands are described in Table 3.1.4; for special selection commands see the following sections:

Select — Group — Section 3.9.2.4 on p. 1281,

Select — Layer — Section 3.5.6.7 on p. 1192,



Select — View — Section 3.5.5.4 on p. 1173,

Select — By Line Style — Section 3.2.1.1.1 on p. 894,








Select — By Attribute — Section 12.3.3.5 on p. 2069,

The objects, from which you want to remove the selection, are specified in the same way you specify objects for selection. The selection removal commands are available when the document has selected objects.

Tab. 3.1.4. General commands for selecting graphic objects

Command	Description
 Select All	Selects all objects contained in the current drawing view (about changing the view state see Section 3.5.5.2 on p. 1171) or in a fragment.
 Select Object	<p>Selects a separate object.</p> <p>After executing the command, point the cursor at the object you want to select.</p> <p>You can pick an arbitrary number of objects to select in a single command call.</p>

Tab. 3.1.4. General commands for selecting graphic objects

Command	Description
	<p>Select by Marquee It serves to select objects of the active document using a rectangular marquee frame.</p> <p>After executing the command, point the cursor at the first and second vertex of the rectangular marquee frame. The elements that are entirely within the set frame will be selected.</p> <p>You can define an arbitrary number of marquee frames in a single command call.</p>
	<p>Select Outside Marquee Serves to select objects of the active document that are outside the specified rectangular marquee frame.</p> <p>After executing the command, point the cursor at the first and second vertex of the rectangular marquee frame. The elements that are entirely outside the set frame will be selected.</p> <p>You can define an arbitrary number of marquee frames in a single command call.</p>
	<p>Select by Cutting Marquee Serves to select objects of the active document that are fully or partially within the specified rectangular marquee frame.</p> <p>After executing the command, point the cursor at the first and second vertex of the rectangular marquee frame. The elements that enter fully or partially into the set frame will be selected.</p> <p>You can define an arbitrary number of cutting marquee frames in a single command call.</p>
	<p>Select by Cutting Polyline Serves to select objects of the active document by crossing them with a polyline.</p> <p>After executing the command, point the cursor at the vertexes of the polyline. The objects intersecting with polyline segments will be selected.</p>
	<p>Select Previous List Serves to select objects that were selected on the previous occasion (elements of the previous list).</p>
	<p>Select by Type Serves to select all objects in the active document according to their type.</p> <p>After executing this command, you will see a dialog where you will have to specify the object types to be selected.</p>
	<p>Select by Properties Selects geometric objects of the active document according to a set of properties, excludes objects from selected objects according to a set of properties, and inverts selection. The Select by Properties command is examined in detail in Section 3.1.3.3.</p>

3.1.3.3. Select Objects by Properties

The **Select by Properties** command allows performing selection operations with objects (geometric, dimensions, designations) matching the specified conditions.

The following selection operations are available:

- ▼ adding objects to a group of selected objects,
- ▼ removing the selection of an object group,
- ▼ inverting the selection of an object group.

Inversion should be understood as selecting all objects in the document (or removing the selection of all objects in the document) with the exception of objects meeting the specified conditions.

The condition for including an object in a group is the correspondence of its properties to specified values. The properties, according to which objects are selected, are parameters that are specified on the Properties bar when creating a specific object. Belonging to types and layers is a common property of all objects.



To call the command, click the button **Select by Properties** on the **Selection** toolbar or select its name in the **Select** menu.

The dialog of object selection by properties will appear (Fig. 3.1.13). The control elements of this dialog allow selecting object types and conditions for performing selection operations on them.

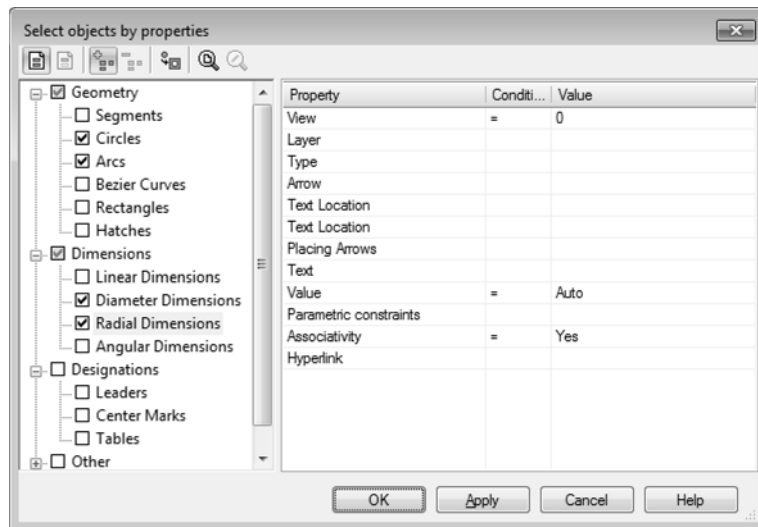


Fig. 3.1.13. Dialog of object selection by properties

The dialog toolbar buttons allow controlling the result of the object selection operations and changing the document view scale.

The object tree contains names of object types present in the current document. This set is formed automatically. The option next to the name allows controlling the inclusion of objects of this type in the group of objects being selected.

The table of selection conditions contains a list of properties of the object selected in the tree and allows specifying the conditions for including the object in the selection group, i.e., forming a selection filter.

To create a filter for selecting objects of a specific type, perform the following actions.

1. In the object tree, enable the option corresponding to the object type.
2. Select the property in the table.
3. In the **Condition** dropdown list, select the required option (Fig. 3.1.14). The set of available conditions depends on the property type.
4. Specify the condition value in the **Value** field. The method of specifying the value also depends on the property type. The values of numerical properties, such as *Length*, *Angle*, or the defining point coordinates must be entered from the keyboard. The values of other properties, such as *View*, *Style*, *Placing Arrows*, are selected from the dropdown list.



Fig. 3.1.14. List of conditions



To quickly specify properties common for all objects in the group, enable a root section option, such as **Geometry**. Select the common conditions and parameter values. In the object tree, disable the options of the object types to be excluded from the group. Proceed to specify the remaining parameters for each object type in the group.

Bear in mind the following specifics when specifying conditions.

- ▼ If the **View** has been selected in the *Property* column and its number has been specified in the **Value** column, when you select a layer the drop-down list will include only layers that belong to this view. If no view has been selected, all layers will be available for selection.
- ▼ If no property value has been specified in the table, the group will include all objects of a given type.
- ▼ If the property values have been specified but the object type option has been disabled, the objects will not be included in the group.
- ▼ If an option in the **Condition** column has not been selected, but the value has been specified, the option = will be used as the condition.
- ▼ If an option in the **Condition** column has been selected, but the value has not been specified, the properties are considered not to have been specified, meaning that the group will include all objects of the given type.
- ▼ If no condition has been specified for a property, no restrictions on selection operations with this object will be applied according to this condition.
- ▼ The group of objects for selection operations includes only those objects of the selected type which satisfy all of the specified conditions.



The buttons **Apply to All Objects** and **Apply to Selected Objects** allow specifying the application range of the filter — all objects in the document or only selected objects. The **Apply to Selected Objects** button is available if the document contains selected objects; it is pressed automatically if the document contained selected objects at the moment the dialog was opened. When this button is pressed, the object tree contains only those object types which are present among the selected objects.



The button **Add to Group** allows adding to the selection group those objects that satisfy the specified conditions.



The button **Exclude from Group** allows excluding from the selection group the previously selected objects that satisfy the specified conditions. The button is available if the document contains selected objects.



The button **Invert Selection** allows inverting the selection state of the objects in the document.

The result of the inversion depends on the status of the **Add to Group** and **Exclude from Group** buttons. If any objects are selected in the object selection tree when the **Add to Group** button is depressed, these objects will be added to the selected objects and the result inverted. When the **Exclude from Group** button is depressed, the selection will be removed from these objects and the result inverted.

After executing the command a single time, you can perform several actions of adding and excluding documents of several types as well as inverting the selection. To perform the action without exiting the dialog, use the **Apply** button.

You can change the document image scale in the following ways without interrupting the command.



The button **Show All** will change the display scale in the active window in such a way as to fully show the entire document.



The button **Selected Objects Scale** serves to automatically change the display scale in the active window in a way as to have all selected objects fit in the largest possible scale. The command is not available if no object is selected.



Selected construction lines and empty (containing no objects) views are ignored when performing the command. A single selected point is also disregarded.

To complete the selection operations, click the **OK** button in the dialog.

After selecting several objects with the selection command, you can execute a different selection command and continue selecting objects — the selection of the previously selected objects will not be removed.

3.1.3.4. Object Picking Over

Sometimes an object you need to specify is located near other objects or overlaps other objects. At the same time, it is difficult (or sometimes utterly impossible) to select it with the cursor.

To select any of the nearby-located objects (including the overlapping ones), use the object pick-over mode. Picking over is possible when the system waits for an object to be **selected** or **specified**, and at the same time several objects fall in the cursor's catcher.

You often need to **select objects** before running a command. For example, to get a copy of an object, you have to select the source object.

Objects must be selected in order to perform certain commands. For example, to construct a parallel line segment you have to specify an object to which the line segment being created should be made parallel.

Picking Over when Selecting Graphical Objects

1. Point the cursor to the group of objects containing the desired one, and select any of them.
2. Call the context menu command **Object Picking Over** or press the `<Ctrl>+<t>` keys.
3. Pick over objects by pressing the `<Space>` key or calling the **Next Object** command from the context menu. Objects grabbed by the cursor catcher at the time of selecting the first object will be highlighted one after another.
4. 4. When the desired object is highlighted, call the **Finish Object Picking Over** command. You can also press the `<Enter>` or left-click the selected object. Picking over will end at the current object.



You do not need to end the picking over. You do not need to end the picking over. You can call the command that requires an object selection right after the latter highlights.

Picking Over Graphic Objects

1. Point the cursor to the group of objects containing the desired object.
2. Without selecting any of them, call the context menu command **Object Picking Over**. You can also press the `<Ctrl>+<t>` keys.
3. Pick over objects by pressing the `<Space>` key or calling the **Next Object** command from the context menu. Objects pointed at by the cursor at the time of calling the picking over command will be highlighted one after another.
4. When the desired object highlights, finish the pick-over mode by confirming the selection. To do this, call the **Select highlighted object** command from the context menu or press the `<Enter>` key. You can also click the mouse on the highlighted object or at any free space of the document window.
5. To exit the picking over mode without selecting any object, call the context menu command **Cancel Picking Over**. You can also press the `<Esc>` key.

The system will return to the mode of executing the command for which the object was selected.

3.1.3.5. Using the Clipboard

When working with drawings, it may become necessary to copy or move some portion of the image within one document or to other documents.

For these purposes, KOMPAS-3D uses its own clipboard, not the standard Windows clipboard.



Sometimes (usually when moving text fragments) the Windows clipboard can be used. It is indicated in the descriptions of the relevant operations.

The amount of information that can be put in the clipboard is unlimited. The clipboard content is saved on disk in a special system file. When copying or cutting to the clipboard again, its previous contents are deleted.

The clipboard helps quickly and conveniently copy or move drawing objects. However, when you need to perform a precise move or complex copying (e.g. on a rectangular or concentric grid), use special edit commands (see Section 3.4.2 on p. 1120 and Section 3.4.3 on p. 1122).



Information can be copied and moved via the clipboard only between documents opened in the same KOMPAS-3D application.

3.1.3.5.1. Placing Objects into Clipboard

Objects can be placed into the clipboard with or without it being cut out from the document.

In the first case, the objects are **cut to the clipboard**. The process that involves cutting to the clipboard and pasting from the clipboard is called **moving via clipboard**.

In the second case, the objects are **copied to the clipboard**. The process that involves copying to the clipboard and pasting from the clipboard is called **copying via the clipboard**.

When copying or cutting to the clipboard again, its previous contents are lost.

To place objects into the clipboard, perform the following actions.

1. Select all objects (for selecting methods see Section 3.1.3 on p. 876) you want to place into the clipboard.



2. Call The **Edit — Cut command** (to cut to the clipboard) or **Edit — Copy** (to copy to the clipboard).



The cursor appearance will change.



3. Select a point that will serve as the base for the selected set of objects.

You do not have to specify a base point, if you have selected:

- ▼ objects belonging to different drawing views,
- ▼ several views in their entirety.

In this case, the origin of the absolute coordinate system is automatically adopted as the base point (the bottom left corner of the drawing sheet).

Also, you do not have to specify a base point if you are placing a text fragment into the clipboard.

3.1.3.5.2. Insert from the Clipboard



To paste objects contained in the clipboard into the active document, call the command **Edit — Paste**.

If the clipboard contains objects belonging to different views of a drawing and/or entire views, they will be immediately pasted into the drawing. The inserted documents are placed in the views with the same parameters as the source objects.



Since a fragment cannot contain views, you cannot paste views or objects belonging to different views into a fragment.

If the clipboard contains objects belonging to a fragment or one kind of drawing view, the ghost of the objects being pasted will be shown on the screen and on the Properties bar — paste control elements. These elements are presented in Table 3.1.5.

Tab. 3.1.5. Clipboard paste control elements

Element	Description
Base point	Coordinates of the base point of the objects being pasted.
Angle of Rotation	Object rotation angle around the base point.
Scale	Object scale factor.
Mode	A group of switches controlling the placement of the objects in multi-layer pasting. Available if the clipboard contains objects belonging to different layers (for layers see Section 3.5.6 on p. 1181). The activation of the To Current Layer switch means the whole image will be placed on one (current) layer of the active document. The activation of the To Source Layers switch means that the missing layers for the object placement will be created in the active document.
Extension Lines	Group of switches controlling the scale of the extension lines and the dimension leaders (see Section 3.4.4.2.1 on p. 1131) *.

* When you copy via the clipboard an object that is a fragment insertion (see Chapter 3.7.2 on p. 1248), the leaders and the extension lines are scaled if the option **Scale Extension Lines** was enabled when pasting this fragment, and not scaled if this option was not enabled, meaning that in this case the status of the **Scaling of Extension Lines** button does not matter.

Once the anchor point is fixed, the clipboard contents are pasted into the document.

You can keep inserting objects, placing them in the appropriate positions within the document, and setting the desired scale and rotation angle. An unlimited number of insertions can be made from the clipboard within one command call.

3.1.4. Grid. Orthogonal drawing

3.1.4.1. Using the Grid

When you work with a graphic document or an operation sketch, it is often very convenient to enable the grid on the screen and establish a link to its nodes. In this case, the cursor will not move smoothly, but discretely through the grid nodes. This working mode can be compared with drawing an image on a square-ruled plotting paper.

The grid is not a part of the document and is not printed to paper.

The grid may appear differently in different windows, even in windows of one and the same document. It is possible to set different grid steps by the respective axes, to draw a grid with nodes, or to assign a grid that is reversed with respect to the current coordinate system or a non-rectangular (skewed) grid.



Management of the grid display in the current window is performed by means of the **Grid** button on the **Current Status** bar.

This button also indicates whether the grid is present in the window: a depressed button means that the grid is enabled, while a released button means it is disabled.

The grid is controlled by means of a special menu that opens when you click the arrow button next to the **Grid** button.

The commands of this menu are described in Table 3.1.6.

Tab. 3.1.6. Grid control commands

Command	Description
Customize parameters	Allows configuring the grid settings in the active window. After you execute the command, the grid setup dialog appears (see Section 9.1.7.2 on p. 1705).
Show parameters	Controls the display of the basic grid parameters — distances between its visible nodes* along the axes X and Y of the current coordinate system — on the Current Status bar (Fig. 3.1.15)**. After you execute the command, the grid parameters will be displayed regardless of whether the grid view is enabled or disabled.
Show Settings if Grid Is On	Controls the display of the basic grid parameters on the Current Status** bar. After you execute the command, the grid parameters will be displayed on the bar only if the grid is enabled.

* When the image is zoomed out, not all grid points are visible on the screen (the grid will be resampled down), which is why the distance between the nearby visible points differs from the distance between the points specified in the grid setup dialog.

If the grid is disabled, the grid parameters under any scale match the grid steps along the axes X and Y specified during setup (see Section 9.1.7.2 on p. 1705).

** The grid parameters are displayed on the bar when it is located horizontally.



Fig. 3.1.15. Displaying grid parameters on the **Current Status** bar

3.1.4.2. Snapping by Grid

The fact that the grid is present on the screen does not mean that the cursor is attached to its points. The activation of the desired snapping option is done separately (see Fig.3.1.3 on p. 868).

The opposite is also true: The opposite is also true: the grid view in the window can be disabled, but this does not prevent the attachment to the grid.

The global snapping to the grid (as any other global snapping) works only in the window in which it was set.

If no constant attachment to the grid is needed, disable the global attachment to the grid.

In this case, you can attach the cursor to a grid node by enabling a local snap (see Section 3.1.1.2.2 on p. 868).

3.1.4.3. Grid View in Zoomed Out Windows

If the grid view in the window is enabled, each time you change the view scale (see Section 1.4.1.2 on p. 76), the system will take into account this change when redisplaying the grid. When the view is zoomed out to a point where the grid with the specified spacing cannot be displayed correctly because of its density, a **resampled down** grid will be displayed.

The grid density limit is determined by the value specified as the **Minimum Grid Spacing** (see Section 9.1.7.2 on p. 1705). As long as the number of pixels between the points exceeds the minimum grid spacing value, the grid is not resampled down. The grid is resampled down when the view is zoomed out beyond this point.

The resampling of the grid is performed taking into account the resampling down step specified during setup.

If the grid spacing divisibility is set to **From Sequence** (see *ibid.*), the grid spacing will be divisible by numbers from the sequence of 2, 5, 10, 20, 50, 100, 200, 500, 1000, 2000, 5000, 10000, 20000, 50000, 100000, 500000. This means that when you set the display scale not allowing to that does not permit correctly displaying every point, there will be displayed every second grid point will be displayed; with further scaling down— every fifth, then tenth, etc.

If the **Ratio** resampling down step has been set (see *ibid.*), the grid spacing will be always divisible by the specified number. For example, setting the resampling-down step a multiple of 2 means that upon decreasing the image scale, first every second of the grid nodes will be shown, then every fourth one, then every sixth, and so on.

3.1.4.4. Orthogonal Drawing

The orthogonal drawing mode is intended for quick creation of objects or their portions that are orthogonal to the axes of the current coordinate system.



Enabling and disabling of this mode can be achieved by clicking the **Orthogonal Drawing** button on the **Current Status** bar. Another way to control the orthogonal mode is to press the *<F8>* key.

This key also serves as an indicator of the orthogonal drawing mode: the pressed button means that the Orthogonal Drawing is enabled, the released button means it is disabled.

If in the process of inputting an object you need to temporarily disable (or enable) this mode, press and hold the *<Shift>* key.

The orthogonal mode is used for drawing horizontal and vertical lines, designations of stepped cuts, mutually perpendicular portions of geometric tolerance branches, and in other cases — form tolerance.

To get familiar with working in the orthogonal drawing mode, complete the following steps.

1. Create a new graphic document (drawing or fragment). Make sure that the orthogonal drawing mode is disabled. Enable it, if necessary.
2. Call the **Tools — Geometry — Lines — Line** and specify the start point of the line with the cursor.
3. Move the cursor around the document field and watch the line segment ghost.
The end point of the line segment coincides with the cursor; it can be fixed at any position on the drawing by clicking the left mouse button.
4. Enable the orthogonal drawing mode.
5. Move the cursor.
Now the line ghost is strictly horizontal or vertical depending on the direction to which the cursor is closer.
6. Set the end point of the segment.
7. Enter the start point of the other line.
8. Move the pointer across the document area. Make sure the line ghost is being constructed orthogonally to the axes of the current coordinate system.
9. Press and hold the *<Shift>* key and continue moving the cursor.
Make sure that the orthogonal mode has switched off, and the segment is being created in the normal mode, following the cursor motion.
10. Release the *<Shift>* button. Make sure that the system is back in orthogonal mode.
11. Create a local coordinate system (see Section 3.1.5.2.1 on p. 888).
12. Create several line segments, selecting their start and end points by the cursor. Make sure that the line segments are created orthogonal to the current LCS axes.

3.1.5. Coordinate Systems in a Graphic Document

3.1.5.1. Absolute coordinate system

Each drawing has an **absolute coordinate system**. Its origin is always in the bottom left corner of the external border. The notion of absolute coordinate system has no meaning for the

fragment (there are no apparent bounds, as in the case of a drawing); therefore, when creating a new fragment, the coordinate system origin is displayed in the center of the window.

3.1.5.2. Local coordinate systems

Using the absolute coordinate system alone is not always convenient. When designing, one often needs to measure distances or corners not from the bottom left corner of the sheet but from a different point.

KOMPAS-3D uses local coordinate systems (LCS) to enable this method of specifying object properties. By setting the LCS at certain points of a part or node being designed, you can quickly select any of them and it current. In this case, all coordinates will be calculated and displayed relative to this coordinate system. When an LCS is no longer needed, you can quickly delete it from the document.

3.1.5.2.1. Creating a Local Coordinate System



To create a local coordinate system in the document, call the command **Insert — Local CS** or press the **Local CS** button on the **Current Status** toolbar.

The LCS control elements will appear on the Properties bar. These elements are presented in Table 3.1.7.

Tab. 3.1.7. The LCS control elements on the Properties bar

Element	Description
Select LCS	A bar containing the list of LCSs in the current document.
Name of the Local CS	Name of the LCS selected in the list. You can enter any name for any LCS in this field.
Origin of the Local CS	Coordinate fields of the origin of the selected LCS. The point coordinates are set up in the current coordinate system.
Slope of the Local CS X Axis	X axis rotation angle field of the selected LCS. This angle is calculated from the X axis of the current coordinate system.
Settings...	Button that lets you configure the way the LCS is displayed. After you click this button, the LCS setup dialog appears (see Section 9.1.7.10 on p. 1712).

3.1.5.2.2. Managing Local Coordinate System



To configure LCSs in the document, make some of them current or remove, call the **Insert — Local CS** command or click the **Local CS** button on the **Current Status** bar.

The elements shown in Table 3.1.7 will appear on the Property bar.



- ▼ To set any LCS as the current coordinate system, select it in the list (when you do so, it will be displayed on the screen) and click the **Current Local CS** button in the **Select LCS** window. A checkmark will appear next to the current LCS in the list.



If none of the LCS is selected as the current one, the absolute coordinate system will be considered the current one.



- ▼ To remove the selected LCS, click the **Delete Local CS** button.
- ▼ In order to create a new LCS, click the **New Local CS** button.

Moreover, selecting any LCS in the list, you can enter a new name, coordinates of the origin, and rotation angle.



You can cancel any of the actions mentioned above by clicking the **Undo** button on the **Standard** bar.

3.1.6. Controlling the Order of Displaying Objects

When displaying and printing graphics documents, the order of drawing objects is taken into account. The default order of drawing is determined by the system automatically depending on the object type: first drawn are hatches and fills, on top of them — auxiliary (construction) lines, then line segments, circles, ellipses, etc. As a result, the objects overlap each other in the order they were drawn.






The overlapping effect is particularly noticeable when working with multicolor fields and hatches as well as with colored lines of enhanced weight.




For example, suppose there are a rectangle, a hatch and a linear dimension in a document. By default, they are ordered as follows: hatch, rectangle, dimension, i.e., the hatch is overlaid by the rectangle and the dimension, while the rectangle is overlaid only by the dimension.

At any time while working with the document, you can modify the default order of the objects with respect to each other. To do this, select the object, whose order needs to be modified, and call the appropriate command from the menu **Edit — Order**. These commands are described in Table 3.1.8.

Tab. 3.1.8. Commands for controlling object visibility

Command	Description
 To Front	Sends the highlighted object on top of the rest. As a result, the relocated object will be on the top, overlapping all the rest.
 To Back	Sends the highlighted object behind the rest of the objects. As a result, the relocated object will be overlapped by all the rest.
 Before Item	Sends the highlighted object before the current one. After calling the command, you need to pick by the cursor the object before which the selected one should be placed. As a result, it will overlap the selected object and all previous objects.

Tab. 3.1.8. Commands for controlling object visibility

	Command	Description
	Behind Object	Sends the highlighted object behind the current one. After calling the command, you need to select the object behind which the selected one should be placed. As a result, it will be overlapped by the selected object and all objects to follow.
	One forward	Swaps the selected object and the object before it.
	Back One	Swaps the selected object and the object behind it.



The visibility control commands are also available in the context menu of the selected object.

The visibility modifying commands can be applied to several selected objects at the same time. In this case, the neighboring objects (those located on the neighboring levels) are moved as a single object.



When you attempt to select with the mouse (see Section 3.1.3.1 on p. 876) one of the overlapping objects, the one in the front will be highlighted. To select objects located under this object, use the pick over command (see Section 3.1.3.4 on p. 881).

When setting up the visibility of the objects in graphic documents, keep in mind of the following special considerations.

- ▼ The visibility can be changed for objects belonging to the current and active layers of a fragment (for details on layers, see Section 3.5.6 on p. 1181) or to the current view of a drawing (for details on views, see Section 3.5.4 on p. 1160).
- ▼ The visibility of the objects belonging to different views is determined by the system automatically: the views that were created more recently are considered in position in front of the views that were created earlier. Therefore, for example, the objects of the very first — the system — drawing view are overlapped by the objects of all the rest of the views. You cannot modify the relative order of the views.
- ▼ The following objects are always in front of all the rest in the drawing objects:
 - ▼ outer and inner frames,
 - ▼ title block,
 - ▼ Specifications,
 - ▼ unspecified surface finish,
 - ▼ BOM in Drawing.
- ▼ Objects forming an embedded fragment (see Section 3.7.2.1 on p. 1248) are arranged relative to one another just as they are arranged in the source fragment. A fragment insertion occupies **one level** of visibility in the main document. Therefore, when working with the main document,

you cannot change the order of the objects that make up the insert. To do this, you need to edit the inserted fragment itself.

- ▼ A macroelement (see Section 3.7.5 on p. 1254), just like a fragment insertion, occupies one visibility level. Immediately after its creation, the macroelement is positioned in front of all the other objects. The objects within the macroelement are positioned with respect to each other in the same way as before the grouping.

3.1.7. Creation of object by sample

One of the methods of creating a new graphical object is creating an object with the properties of an existing one (sample). In order to create an object similar to an existing one, use the **Object by sample** command.

When performing the **Object by sample** command, properties of the sample object are copied to the new object. Samples may be geometrical objects (except for contours and elliptical arcs), symbols, and dimensions. The set of copied properties depends on the sample object type.

Example. There is a fill inside a circle in a document. The **Object by sample** command makes it possible to create another fill with the same properties (type, color, transparency, and so on) inside a different contour.

In order to create an object by sample:



1. Click the **Object by sample** button on the **Geometry** toolbar (see Fig. 3.2.1) or select its name in the **Instruments** menu.



The cursor will change its appearance.

2. Specify the object that will be used as a sample for copying properties. For this purpose, place the cursor on the object. When the selected object highlights, click the left mouse button.

Once the object is specified, the command, corresponding to the specified object's type, will be run. For example, if the sample object is a segment, the **Segment** command will be run.

3. Specify the parameters of the new object. For example, segment requires vertices to be specified. Style of the segment's line will be the same as the style of the sample segment.
4. Finish creating the object.

The **Object by sample** command can be interrupted prior to running the command, corresponding to the specified object's type. You can do it using one of the following methods:



- ▼ Click the **Interrupt command** button on the Special Control Panel,
- ▼ Call the **Interrupt command** command from the context menu,
- ▼ Press the <Esc> key.



If the sample objects has already been highlighted prior to calling the **Object by sample** command, the command corresponding to the sample object type will be run immediately after calling the command.

3.2. Geometrical Objects

3.2.1. General Information on Geometric Objects

It is possible to create geometrical objects of the following types in KOMPAS-3D system:

- ▼ points,
- ▼ lines,
- ▼ line segments,
- ▼ circles,
- ▼ ellipses,
- ▼ circle arcs and ellipses,
- ▼ rectangles,
- ▼ regular polygons,
- ▼ polylines,
- ▼ splines (NURBS, Bezier curves),
- ▼ multilines,
- ▼ hatches and fills,
- ▼ offset curves,
- ▼ contours.

Commands for creating these objects are grouped under the **Tools — Geometry** menu, and buttons for executing the commands — on the **Geometry** bar (Fig. 3.2.1).



Fig. 3.2.1. **Geometry** bar

If any actions are performed identically for similar commands, such actions will be described only once when you first come across them in this text.

3.2.1.1. Geometric Object Styles

The appearance of a geometric object is determined by its **style**.

KOMPAS-3D comes with system styles for points, lines, and hatches. System styles of lines and hatches correspond to standard styles. You can create custom styles for lines and hatches.

The *Auxiliary* style is one of the system styles for points and lines. It is intended for objects that perform auxiliary functions— points and lines created during layout operations, preliminary design, etc. Once these objects are no longer necessary, they are deleted.

If auxiliary objects have the same style, you can quickly delete all of them by clicking **Edit—Delete— Construction Curves and Points** (see Section 3.4.8.1 on p. 1144). This is why we recommend using a corresponding style when building an auxiliary object. If the object you are

creating is not going to be deleted together with auxiliary objects, you should select a different style for this object.

When you are creating a geometric object, the current style is shown in the same-titled field on the Property bar. To change the style, open the **Style** list and select the required style name (Fig.3.2.2).

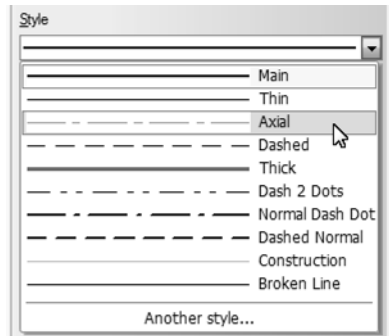


Fig. 3.2.2. Selecting the line style

The list and sequence of line styles in the **Style** list as well as the default style are determined by the line style list setting (see Fig.9.2.6.6 on p. 1763).



When you create lines (see Section 3.2.3 on p. 901), they are automatically assigned the *Auxiliary* style. It is impossible to choose a different style.

The appearance of points and lines with system styles depends on the system settings. If necessary, you can modify it (see Sections 9.1.7.4 on p. 1707 and 9.1.7.5 on p. 1708).

3.2.1.1.1. Selecting Curves by Style

To select all same-style lines in the document, perform the following steps.



1. Call the **Select — By Line Style...** command. A dialog listing all line styles used in the document will appear.
2. Select the required style(s) in the dialog.
Objects having the specified style(s) will be selected.

3.2.1.1.2. Changing the Object Style

To change the style of an existing geometric object(s), perform the following steps.

1. Select the object(s) whose styles you want to change (for selection methods see Section 3.1.3 on p. 876).
2. Call the **Service— Change Style** command. A style modification dialog will appear (Fig.3.2.3). The number of dialog tabs depends on the type of selected objects (lines, points, hatches, text).
3. Specify the settings and confirm style change.
See section 3.6.2.7.1 on p. 1220 for more information about changing the style of hatches in associative views of a drawing.



Fig. 3.2.3. Style change dialog

3.2.1.2. List of line styles

Every graphic document stores a **list of line styles**: the information on what styles can be used in this document.

By default, the list contains system line styles only. If necessary, the user can add styles to the list from style libraries or sets.

You can apply the **style filter** to the line style list to limit the set of styles displayed in the **Style** list on the Property Bar (see Fig.3.2.2) when creating geometric objects. It is recommended to show several most used styles only. When needed, you can choose any other style **or create a new one** to draw an object using the **Another Style...** row in the Style list. Line style filter settings are described in Section 9.2.6.6.1 on p. 1763.

3.2.1.2.1. Line Style Manager

The Line Style Manager (Fig.3.2.4) serves to manage a line style list of the current or new documents.

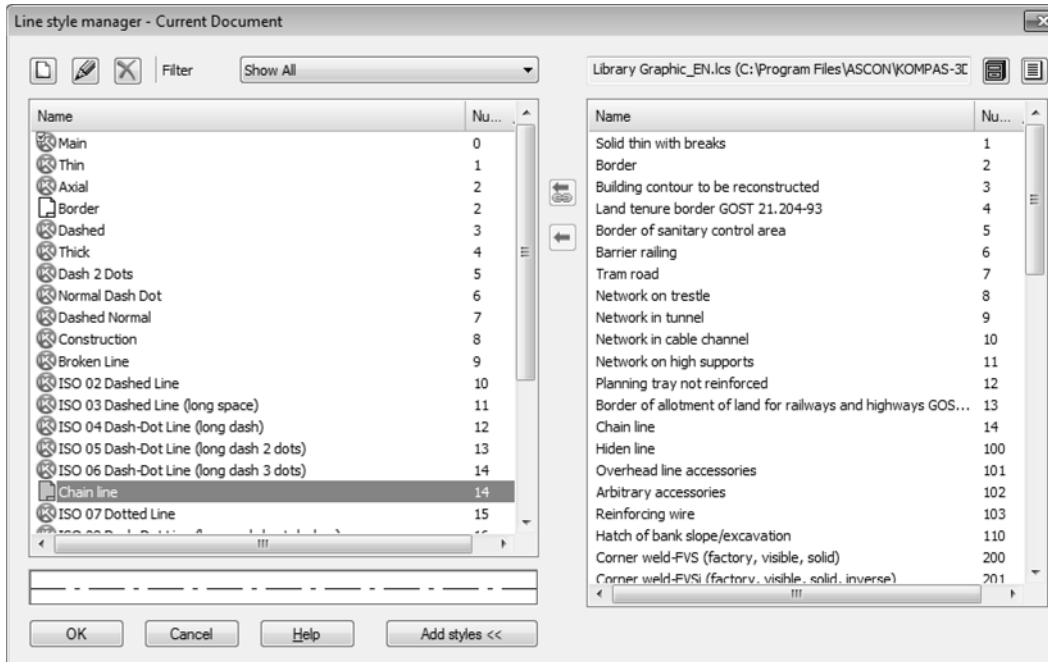


Fig. 3.2.4. Line Style Manager during the line style list setting up for the current document

By default, the Line Style Manager is opened in minimized view, i.e.

only its left part is displayed. It contains the list of line styles: The styles are marked with the following signs:




- ▼ **System** style,
- ▼ **Embedded** in a document style,
- ▼ **Library** style.

In the line style list of the current document the marks can be accompanied by a “check” showing that the style is involved in the document, i.e. is **used**. For instance, Figure 3.2.4 shows the *Main* and *Auxiliary* styles that are used.






The right part of the Line Style Manager appears after clicking the **Add Styles** button. This part serves for selection of a library (set) of styles and embedding of styles from it to the list of document styles.

The Line Style Manager controls are described in Table 3.2.1.


Tab. 3.2.1. Line Style Manager Controls

Element	Description
 New	The button allowing creation of a style. A new style is added to the document style list as an embedded one. The curve style creating or editing dialog is displayed after this button is clicked.

Tab. 3.2.1. Line Style Manager Controls

	Element	Description
	Edit	<p>The button making it possible to edit a selected style. It is available when an embedded style is selected in the document style list.</p> <p>The curve style creating or editing dialog is displayed after this button is clicked.</p>
	Delete	<p>The button allowing to delete selected styles. It is available if unused styles (embedded and/or library ones) are selected in the document style list.</p> <p>After clicking this button, the dialog that allows to confirm or revoke the deletion.</p>
	Filter	<p>List of filters for the list of document styles:</p> <ul style="list-style-type: none"> ▼ Show All, ▼ System Styles, ▼ Embedded Styles, ▼ Library Styles, ▼ Used Styles, ▼ Unused Styles. <p>To apply a filter, select it from the Filter list. Only those styles, which correspond to the applied filter, will remain in the list.</p>
	Add styles	<p>This button allows to add styles from a library or set to the document style list. After clicking it an additional area with control elements will appear in the Line Style Manager.</p> <p>To close this area, the Add Styles button shall be clicked repeatedly.</p>
	Show Library	<p>The button serves for line style library selection. After its clicking, the dialog where it is required to indicate the line style library file (*.lcs) and click the Open button, appears on the screen.</p> <p>The list of library styles of lines appears in the preview window.</p>
	Show Set	<p>The button serves for line style set selection. After its clicking, the dialog where it is required to indicate the set name and click the Select button, appears on the screen.</p> <p>The list of line styles from a set appears in the preview window.</p>
	Add library style	<p>It allows to add one or several library styles to the document style list. It is available when there are selected styles in the list of library styles.</p> <p>After clicking this button, the selected styles will be included into the document style list as the library styles.</p> <p>Library styles maintain the link with library files, from which they were added to the document.</p>

Tab. 3.2.1. Line Style Manager Controls

Element	Description
	<p>Embed Style into Document</p> <p>It allows to embed one or several library styles or styles from a set to a document.</p> <p>It is available when there are selected styles in the list of library styles or styles form a set. After clicking this button, the selected styles will be included into the document style list as the embedded styles.</p> <p>Styles embedded to a document from libraries do not maintain the links with the files of these libraries.</p>

Library styles with broken connection

Document library styles lose connections with the library in the following cases:

- ▼ you move the library file from which the styles were added to the document to another folder;
- ▼ you delete respective styles from the library.

When you open the document which style list contains styles with broken connection, the message pops up saying that styles or style libraries are not found.



Library styles with broken connection have an exclamation mark within a red circle in the Line Style Manager.

Connections can be restored in one of the following ways:

- ▼ place the library file using the previous path and then close and open the document— the library and its styles will be found;
- ▼ without moving the library, open it in the Line Style Manager and add its styles to the document style list once more— the previous library file path will be replaced with the current one.



If there are several library styles with broken connection in the list, you can use the **Library, with Broken Connection** filter for convenience in operation.

If a library or its styles cannot be restored, edit the objects that use styles with broken connection by setting other styles for them and then delete the styles with broken connection from the document style list. For instance, you can do the following.

1. Open the Line Style Manager, remember the style name that lost connection with the library and close the Line Style Manager.
2. Select the document objects that use this style with the **Select— By Line Style...** command.
3. Replace the style of the selected objects with another one (see Section 3.2.1.1.2 on p. 894).
4. Open the Line Style Manager, delete the style that lost connection with the library and close the Line Style Manager.



The lines that use the style with broken connection retain their typeface, therefore you may leave their style unchanged (e.g. if there are no styles to be replaced). But you won't be able to delete the style with broken connection in this case; a warning will pop up when the document is opened.

3.2.2. Points

KOMPAS-3D offers a variety of ways to draw points as well as several point layout styles. This allows using points both as auxiliary elements and independent geometric objects.

3.2.2.1. Random Point



To draw a randomly located point, click **Point**.
Define the point position.

3.2.2.2. Points on Curve



To draw several points dividing a curve into several equal sections, click **Points on Curve**.
In the relevant field on the Property bar, enter the number of sections that you want the curve divided into.

Select a curve to set points.

If a curve is not closed, then the points will be created right after its selection. The first point will coincide with the start point of the curve, the last point—with the end point.

If a curve is closed, then, after selecting it, you need to define the position of the first point on it (define the point **p1**).



Fig. 3.2.5. Example of curves (closed and open) divided into equal sections



If the specified point does not belong to the selected curve, then the first point position will be determined by the projection (mapping) of the specified point on the curve. Use snaps for precise positioning of the cursor (see Section 3.1.1 on p. 865).



If you want to break a curve into several equal sections without drawing points, use the **Break Curve into N Parts** command (see Section 3.4.6.2 on p. 1136).

3.2.2.3. Intersection Points of Two Curves



To draw points at intersections of curves, click **Intersection Points of Two Curves**.

Select a curve to search for intersections. Then, sequentially select the curves that intersect it.

After selecting each subsequent curve, the points are automatically created in the locations of its intersection with the first curve.

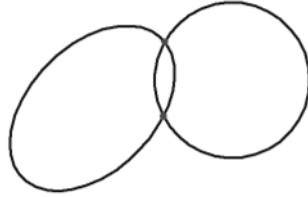


Fig. 3.2.6. Example of drawing points at intersections of an ellipse and circle



To select another line to process for intersections, click the button the **Reselect** button and proceed to specify the required lines.

3.2.2.4. All Intersection Points of Curve



To draw points at all intersections of the selected curve with other lines, click **All Intersection Points of Curve**.

Select a curve to search for intersections.

Points will be created automatically at its intersections with other curves located in current and active views (for drawing views see Section 3.5.5.2 on p. 1171) and layers (for layer types see Section 3.5.6 on p. 1181).



Fig. 3.2.7. Example of setting all intersection points of an ellipse with two curves



You can draw all intersection points of an auxiliary line immediately after creating it using the **Mode** switch (see Section 3.2.3.1.1 on p. 901).

3.2.2.5. Point at Specified Distance



To draw points on a curve located at a specified distance from a selected point on this curve (reference point) and from each other, click **Point at Specified Distance**.

In the relevant field on the Property bar, enter the number of points you want to create.

Select a curve to set points.

The **Reselect** button allows selecting a different curve for drawing points.

Specify the base point on a curve— the **p** point.



If the specified point does not belong to the selected line, then the base point position will be determined by the projection (mapping) of the specified point on the line. Use snaps for precise positioning of the cursor (see Section 3.1.1 on p. 865).

In the **Distance** field of Properties bar, enter the distance between the reference point in the first point being created. If several points are being created, then the specified value will also define the distance between them.

As the cursor moves around the reference point, the ghosts are displayed for the point (points) that can be created. Fix the required ghost.



Fig. 3.2.8. Example of three points drawn along an arc at a distance of 5 mm from the reference point

3.2.3. Construction lines

Straight lines are the equivalent of thin lines used by the designer on the drawing board. They are required for preliminary construction that is subsequently used to draw the final contour of a part, and sometimes— to specify projection correlations between views.

Straight lines have the *Auxiliary* style that cannot be changed.

Auxiliary straight lines (as well as other lines with the *Auxiliary* line style) are not printed out to paper.

3.2.3.1. Arbitrary Straight Line



To draw an arbitrary straight line, click **Auxiliary Line**.

Specify the first point through which the straight line should pass— **p1**.

- ▼ If you know the location of the second point belonging to the straight line (**p2**), specify it. The line inclination angle (the angle between the line and the X-axis of the current coordinate system) will be determined automatically.

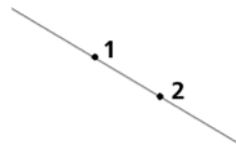


Fig. 3.2.9. Creating a line by two points

- ▼ If you know the tilt angle of the straight line you are creating, enter it into the relevant field on the Property bar.

3.2.3.1.1. Drawing Intersection Points

When building auxiliary elements, you sometimes need to draw points where lines intersect between each other and with other objects.



By default, these points are disabled, and the **Do not Set Intersection Points** switch is active in the **Mode** group. To enable the points, activate the **Set Intersection Points** switch.

If the mode of setting intersection points is enabled, then, when creating a new auxiliary line, the system will automatically put intersection points of that line with all graphical objects lying on the active layers of the current view (See Section 3.5.5.1 on p. 1171).

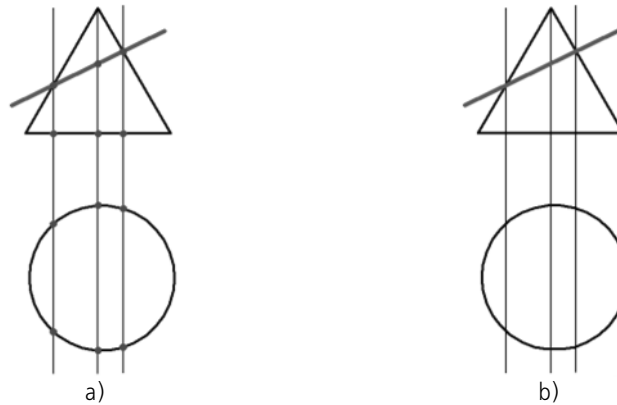


Fig. 3.2.10. Creating auxiliary lines a) with auxiliary points, b) without auxiliary points

Automatically drawn points have the *Auxiliary* style.

If necessary, you can delete them all by a single command together with other auxiliary (construction) elements (see Section 3.4.8.1 on p. 1144).



If you need to select not all intersection points on the line, but only its points of intersection with certain objects, use the **Intersection Points of Two Curves** command (see Section 3.2.2.3 on p. 899).

3.2.3.2. Horizontal Line



To create a horizontal straight line, click **Horizontal Line**.

Then define the point, through which the created line should go.



A line which is parallel to the X-axis of the current coordinate system is a horizontal line. Therefore, if you create a horizontal line on a view, whose coordinate system is rotated relative to the absolute coordinate system, then this line will not be parallel to horizontal borders of the sheet.

3.2.3.3. Vertical Straight Line



To create a vertical straight line, click **Vertical Line**.

Then define the point, through which the created line should go.



A line which is parallel to the Y-axis of the current coordinate system is a vertical line. Therefore, if you create a horizontal line on a view, whose coordinate system is rotated relative to the absolute coordinate system, then this line will not be parallel to vertical borders of the sheet.

3.2.3.4. Parallel Line



To draw a line parallel to a rectilinear object, click **Parallel Line**.

Rectilinear objects include:

- ▼ construction lines,
- ▼ line segments (also as a part of polylines, polygons, contours),
- ▼ dimension lines and dimension leaders,
- ▼ table lines.

Specify a straight object relative to which you want to draw a parallel line.

Specify the distance from the straight object to the parallel line. You can do this by the following methods:

- ▼ enter the distance value into the relevant field on the Property bar,
- ▼ then define the point, through which the created line should go.

By default, the system shows the ghosts of two lines positioned at the specified distance on either side of the straight object. In this case, the **Two Straight Lines** switch is active in the **Number of Straight Lines** group.



To enable the creation of a single line parallel to the object, activate the **One Straight Line** switch.

Fix the ghost of a line (or two lines).

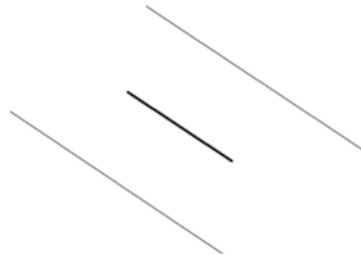


Fig. 3.2.11. Lines parallel to a line segment



To proceed to building lines parallel to a different object, click the **Reselect** button and proceed to select the required straight object with the pointer.

3.2.3.5. Perpendicular Straight Line



To build a straight line perpendicular to another object, click **Perpendicular Line**.

Select the object, which the straight line should be tangent to. Proceed to specify a point on the straight line.

The ghosts will be displayed for all possibilities of straight lines perpendicular to the selected object and going through the specified point.

Activate a suitable ghost and fix it.



Fig. 3.2.12. Lines perpendicular to the spline



To proceed to building lines perpendicular to a different object, click the **Reselect** button and select the required object.

3.2.3.6. Straight Line Tangent to Outside Point



To draw a line tangent to an object and passing through a point specified outside of this object, click **Straight Line Tangent to Outside Point**.

Select the object to which the straight line should be tangent. Proceed to specify a point on the straight line.

The ghosts of all possible lines tangent to the object and passing through the specified point will be displayed.

Activate a suitable ghost and fix it.

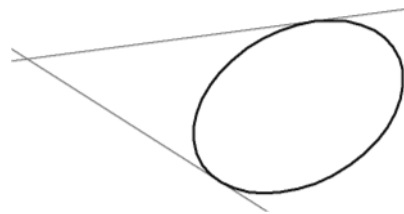


Fig. 3.2.13. Straight lines tangent to ellipse

To proceed to building lines tangent to a different object, click the **Reselect** button and proceed to select a different object.

3.2.3.7. Straight Line Tangent to Point on Curve



To draw a line tangent to an object at point on the object, click **Straight Line Tangent to Point on Curve**.

Select the object to which the straight line should be tangent. The arc ghost will be displayed.

- ▼ If you know the tangent point location, specify it.



If the specified point does not belong to the selected line, then the dividing point position will be determined by the projection of the specified point on the line. Use snaps for precise positioning of the cursor (see Section 3.1.1 on p. 865).

- ▼ If you know the inclination angle of the tangent line (the angle between the line and the X-axis of the coordinate system), enter it in the **Angle** field on the Property bar.
Fix the ghost(s).



Fig. 3.2.14. Line tangent to the spline: a) passing through a spline point, b) having a specified tilt angle



To proceed to creating straight lines tangent to another object, click the **Reselect** button and proceed to specify the required object.

3.2.3.8. Straight Line Tangent to Two Curves



To create a line tangent to two objects, call the **Straight Line Tangent to Two Curves command**.

Select the first and second objects with the cursor, to which the line should be tangent.

The ghosts of all possible tangency cases will be displayed.

Activate a suitable ghost and fix it.

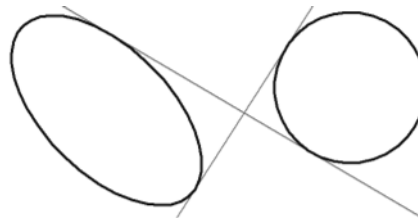


Fig. 3.2.15. Straight lines tangent to ellipse and circle

To proceed to building lines tangent to other objects, click the **Reselect** button and proceed to select two objects.

3.2.3.9. Bisector



To build a bisector of an angle formed by two selected rectilinear objects, click **Bisector**.

Select any two rectilinear objects one by one.

The ghosts will be displayed of the bisectors of the angles made up by the selected objects.

Activate a suitable ghost and fix it.

If you select two parallel objects, the system will build a line equidistant from these objects.



To select other objects for building a bisector, click the **Reselect** button and proceed to select two rectilinear objects one by one.

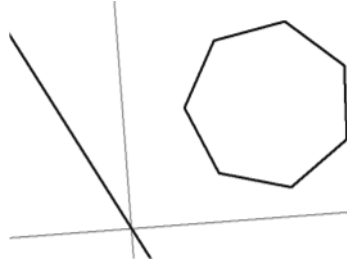


Fig. 3.2.16. Bisectors of angles formed by a line segment and a polygon side

3.2.4. Line Segments

3.2.4.1. Line Segment



To draw an arbitrary line segment, click **Line Segment**.

Specify the start point of the line segment **p1**.

- ▼ If you know the location of the end point of the line **p2**, specify it. In this case, the length and inclination angle of a line segment will be determined automatically.
- ▼ If you know the length and tilt angle of the line segment, enter them in the relevant fields on the Property bar. The location of the end point will be defined automatically.

3.2.4.2. Parallel Line Segment



To draw a line parallel to a rectilinear object, click **Parallel Line Segment**.

Select an object to which the line segment should be made parallel.

Specify the start point of the line segment **p1**. The distance from the object to the parallel line segment will be determined automatically.



You can enter this distance in the relevant field on the Property bar before specifying the location of the start point of the line segment.

- ▼ If you know the location of the end point of the line **p2**, specify it.
- ▼ If the line segment length is known, enter its value in the Length field on the Property bar.

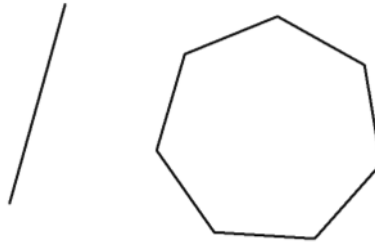


Fig. 3.2.17. Creating a line segment parallel to a rectangle side



To proceed to creating line segments parallel to another object, click the **Reselect** button and proceed to specify the required object.

3.2.4.3. Perpendicular Line Segment



To build a line segment perpendicular to a rectilinear object, click **Perpendicular Line Segment**.

Select an object to which the line segment should be made perpendicular.

Specify the start point of the line segment **p1**.

- ▼ If you know the location of the end point of the line **p2**, specify it.
- ▼ If the line segment length is known, enter its value in the Length field on the Property bar.

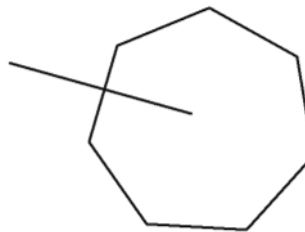


Fig. 3.2.18. Building a line segment perpendicular to a polygon side



To proceed to building line segments perpendicular to a different rectilinear object, click the **Reselect** button and proceed to select the required object.

3.2.4.4. Line Segment Tangent to Outside Point



To draw a line segment tangent to an object, click **Line Segment to Outside Point**.

Select the line to which the line segment should be tangent.

Then select the segment start point **p1** off the curve.



By default, the line end point is the tangency point. If necessary, you can modify the line segment length by entering the appropriate value in the respective field on the Property bar.

The ghosts of all possible line segments tangent to the given object and passing through the point **p1** will be displayed.

Activate a suitable ghost and fix it.

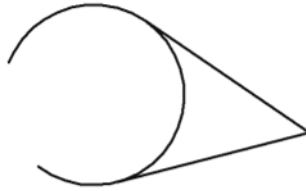


Fig. 3.2.19. Building line segments tangent to an arc



To proceed to building lines tangent to another object, click the **Reselect** button and proceed to select the required object.

3.2.4.5. Line Segment Tangent to Point on Curve



To draw a line segment tangent to an object and passing through a specified point on this object, click **Line Segment Tangent to Point on Curve**.

Select the object to which the line segment should be tangent. The arc ghost will be displayed.

- ▼ If you know the tangent point location, specify it.

And the value of the inclination angle and **p2** point coordinates of the tangent segment are determined automatically.

- ▼ If you know the inclination angle of the tangent line segment (the angle between it and the X-axis of the coordinate system), enter it in the **Angle** field on the Property bar.

Once the angle value is entered, ghosts for all variants of tangents positioned at a certain angle are displayed. Choose the appropriate variant of the tangent. For this purpose, move the cursor to the desired ghost. Once its respective tangent point appears on the screen, fix it by clicking on it with the mouse.

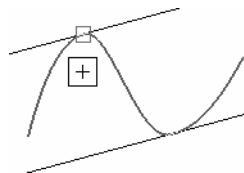


Fig. 3.2.20. Selection of tangent point

Select the point **p1** defining the segment length. If the tangent line segment length is known, enter its value in the **Length** field on the Properties bar.



Use snaps to position the cursor precisely (see Section 3.1.1.3 on p. 870) or the geometric calculator (see Section 3.1.2 on p. 871).

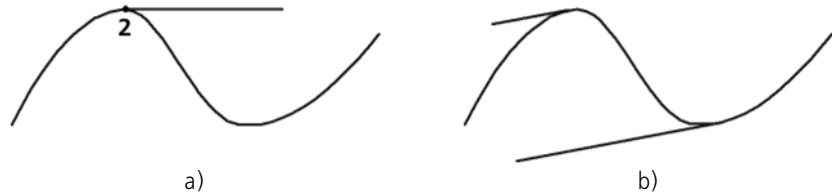


Fig. 3.2.21. Creating a tangent line segment:
a) specifying the tangent point, b) specifying the tilt angle

3.2.4.6. Line Segment Tangent to Two Curves



To draw a line segment tangent to two curves, click **Line Tangent to Two Curves**.
Select the first and the second curve, to which the line segment should be tangent.



By default, the system offers to create line segments with the end points in the tangency locations. If necessary, you can enter the desired line segment length value in the respective field on the Property bar.

The ghosts will be displayed for all possible segments tangent to the specified curves.
Activate a suitable ghost and fix it.

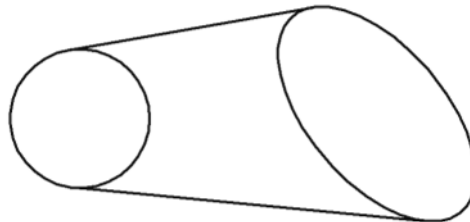


Fig. 3.2.22. Line segments tangent to an ellipse and circle



To proceed to building line segments tangent to other objects, click the **Reselect** button and proceed to select two curves with the cursor.

3.2.5. Circles

3.2.5.1. Circle



To draw an arbitrary circle, click **Circle**.
Select the circle centerpoint.

- ▼ If you know the location of the point **p** through which the circle passes, specify this point.

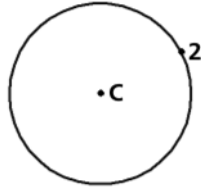


Fig. 3.2.23. Creating a circle by center and point



▼ If you know the value of the circle radius or diameter, enter it in the relevant parameter field on the Property bar. The system awaits diameter entry by default. In the **Parameter** group, the **Diameter** switch is active.



To enter the circle radius, activate the **Radius** switch.

3.2.5.1.1. Circle with Axes



By default, circles are built without axis lines. In this case, the **No Axes** switch is active in the **Axes** group on the property panel.



For the circle you are creating to have axis lines, activate the **With Axes** switch. Axes drawn along the directions of the current coordinate system will appear on the circle ghost.

Axes are created similarly when you build circles of other types.



If you have drawn a circle without axes but then discovered you need the axes as well, you do not need to delete the circle and create a new one with axes. Enter the circle editing mode by double-clicking the circle. Enable the drawing of axes and click the **Create Object** button on the Advanced Control panel.

You can also build axes using the **Center Mark** command (see Section 3.3.7.14 on p. 1048).



Axis lines represent a system macro element—the center mark. By default, the center mark is not linked with the circle and is not rebuilt during subsequent editing of the circle (changes made to the radius or location).

3.2.5.2. Circle by Three Points



To draw a circle passing through three specified points, click **Circle by Three Points**.

Specify the points **p1**, **p2** and **p3**, through which the circle should pass. The circle center coordinates and its radius will be determined automatically.

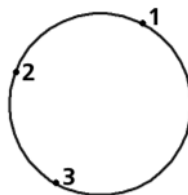


Fig. 3.2.24. Circle by Three Points

3.2.5.3. Circle with Centerpoint on Object



To draw a circle with a centerpoint on a specified curve, click **Circle Centerpoint on Object**. Select the object, to which the centerpoint of a circle should belong.

Specify the first point **p1** through which the circle you are creating will pass.

- ▼ If you know the location of the second point **p2** on the circle, specify it.



- ▼ If you know the value of the circle radius or diameter, enter it in the relevant parameter field on the Property bar. The system awaits diameter entry by default. In the **Parameter** group, the **Diameter** switch is active.



To enter the circle radius, activate the **Radius** switch.

The ghosts of all circle configurations satisfying the specified parameters will appear on the screen.

Activate a suitable ghost and fix it.

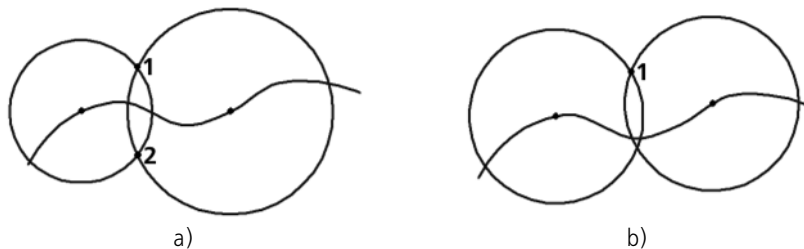


Fig. 3.2.25. Circles with centerpoints on the spline: a) passing through points **p1** and **p2**, b) with equal radii and passing through point **p1**



To proceed to building circles with a centerpoint on a different object, click the **Reselect** button and proceed to select a new object with the pointer.

3.2.5.4. Circle Tangent to One Curve



To draw a circle tangent to a selected curve, click **Circle Tangent to Curve**.

Select an object to which the circle should be made tangent.

- ▼ If you know the location of points **p1** and **p2** belonging to the circle you are creating, specify them (Fig. 3.2.26, a).



You can enter the circle radius or diameter in the relevant field on the Property bar before specifying the second circle point. However, building a tangent circle is possible not with all combinations of the point location on the circle (**p1**) and the radius value. If the circle ghost disappears after you enter the radius value, this means that building the circle is impossible.

- ▼ If you know the point of the circle center, enter it (Fig.3.2.26,b).

Ghosts of all circles with specified parameters and tangent to the selected curve will appear on the screen.

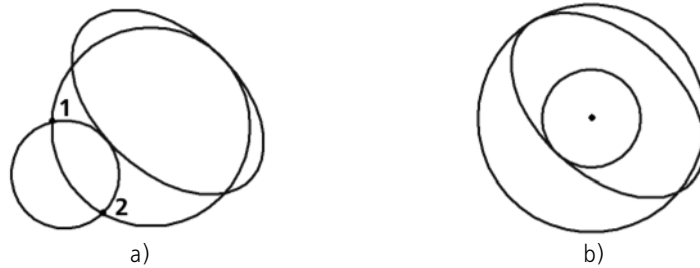


Fig. 3.2.26. Circles tangent to ellipse:
a) passing through points **p1** and **p2**, b) concentric

Activate a suitable ghost and fix it.



To proceed to building circles tangent to another object, click the **Reselect** button and proceed to select a new object with the pointer.

3.2.5.5. Circle Tangent to Two Curves



To draw a circle tangent to two selected curves, click **Circle Tangent to Two Curves**.

Select objects to which the circle should be made tangent.

- ▼ If you know a point **p** belonging to the circle you are creating, specify it.
- ▼ If you know the value of the circle radius or diameter, enter it in the relevant parameter field on the Property bar. The system awaits diameter entry by default. In the **Parameter** group, the **Diameter** switch is active.



To enter the circle radius, activate the **Radius** switch.

Ghosts of all circles with specified parameters and tangent to the selected curves will appear on the screen.

Select the intended ghost and fix it.

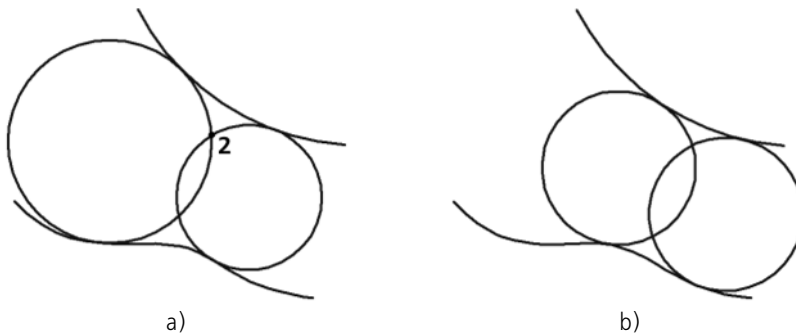


Fig. 3.2.27. Circles tangent to arc and spline:
a) passing through point **p**, b) of equal radii



To proceed to building circles tangent to other objects, click the **Reselect** button and proceed to select two objects with the pointer.

3.2.5.6. Circle Tangent to Three Curves



To draw a circle tangent to three selected curves, click **Circle Tangent to Three Curves**. Select the first, the second and the third object to which the circle should be tangent.



If there is an ellipse or spline among the selected objects, then you will need to specify an approximate circle position.

The ghosts will be displayed for all possible circles tangent to the specified objects. Activate a suitable ghost and fix it.

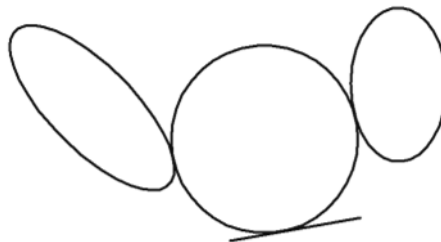


Fig. 3.2.28. Circle tangent to two ellipses and a line segment

To proceed to creating circles tangent to other objects, click the button **Reselect** on the Advanced Control Panel, and then select sequentially three objects using the cursor.

3.2.5.7. Circle by Two Points



To create a circle going through two specified points, click **Circle by Two Points**.

Specify the first point **p1** through which the circle will pass.



Enter the diameter or radius in the parameter entry field on the Property bar, when the **Diameter** or **Radius** switch is active.



Specify the second point **p2** through which the circle will pass.



If the points **p1** and **p2** are diametrically opposite, you do not have to enter the radius or diameter— it will be calculated automatically.

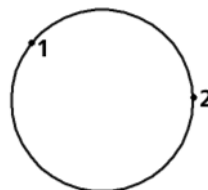


Fig. 3.2.29. Creating a circle by two points

3.2.6. Ellipses

When you build ellipses using all of the commands described below, the drawing of axis lines is controlled in the same way as when you are creating circles (see Section 3.2.5.1.1 on p. 910).

3.2.6.1. Ellipse



To draw an arbitrary ellipse, click **Ellipse**.

Specify the ellipse centerpoint.

Specify the value of the first ellipse semiaxis. This can be completed by the following methods.

- ▼ Specify the end point of semiaxis **p1**.
 - ▼ enter the semiaxis length into the relevant field on the Property bar.
- The tilt angle of the first semiaxis to the X axis of the current coordinate system is determined automatically.



If necessary, before specifying the value of the first semiaxis, you can specify the exact value of its tilt angle in the relevant field on the Property bar.

Specify the value of the second ellipse radius. You can also do this in two ways.

- ▼ Specify the end point of semiaxis **p1**.
- ▼ enter the semiaxis length into the relevant field on the Property bar.

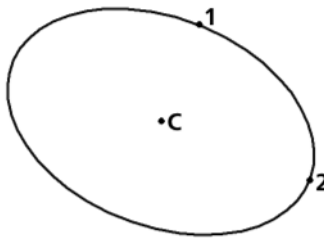


Fig. 3.2.30. Ellipse by centerpoint and semiaxis dimensions

3.2.6.2. Ellipse by Diagonal of Circumscribed Rectangle



To draw an ellipse inscribed in a rectangle with a specified diagonal, click **Ellipse by Diagonal of Rectangle**.

In the respective field of the Property bar, enter the value of the ellipse's first semiaxis inclination angle with respect to the X-axis of the current coordinate system (by default, this value is 0°).

Then, select the start (**p1**) and end (**p2**) points of the diagonal of the rectangle circumscribed about the ellipse being created. The lengths of the ellipse's semi-axes will be calculated automatically.

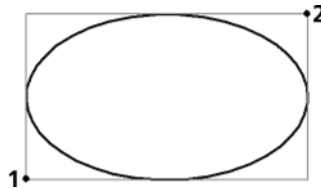


Fig. 3.2.31. Ellipse by diagonal of circumscribed rectangle

3.2.6.3. Ellipse by centerpoint and vertex of circumscribed rectangle



To draw an ellipse inscribed in a rectangle with a specified centerpoint and vertex, click **Ellipse by Centerpoint and Vertex of Rectangle**.

In the respective field of the Property bar, enter the value of the ellipse's first semiaxis inclination angle with respect to the X-axis of the current coordinate system (by default, this value is 0°).

Then, select the centerpoint and a vertex of the rectangle circumscribed about the ellipse being created. The lengths of the ellipse's semi-axes will be calculated automatically.

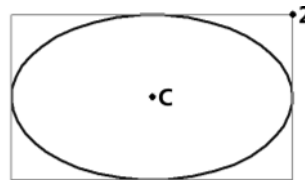


Fig. 3.2.32. Ellipse by centerpoint and vertex of circumscribed rectangle

3.2.6.4. Ellipse by Centerpoint, Midpoint of Side and Vertex of Circumscribed Parallelogram Command



To draw an ellipse inscribed in a parallelogram with a specified centerpoint, midpoint and vertex, click **Ellipse by Centerpoint, Midpoint of Side and Vertex of Parallelogram**.

Specify the center point, and then the mid-side (**p1**) and the vertex (**p2**) of the parallelogram circumscribed about the ellipse being created.

The lengths of an ellipse's semi-axes and the inclination angle of its first semi-axis with respect to the X-axis of the current coordinate system will be calculated automatically.

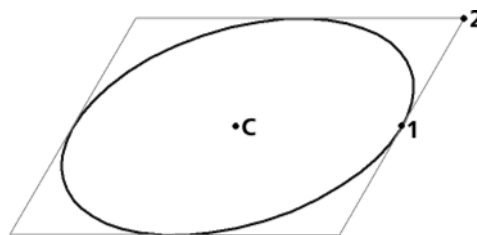


Fig. 3.2.33. Ellipse by centerpoint, midpoint of side and vertex of parallelogram

3.2.6.5. Ellipse by Three Vertices of Circumscribed Parallelogram



To draw an ellipse inscribed in a parallelogram with three specified vertices, click **Ellipse by Three Parallelogram Vertices**.

Specify the positions of the vertices **p1**, **p2** and **p3** of the parallelogram circumscribed above the ellipse being created.

The lengths of an ellipse's semi-axes and the inclination angle of its first semi-axis with respect to the X-axis of the current coordinate system will be calculated automatically.



Fig. 3.2.34. Ellipse by 3 vertices of circumscribed parallelogram

3.2.6.6. Ellipse by Centerpoint and Three Points



To draw an ellipse with a specified centerpoint and passing through three specified points, click **Ellipse by Centerpoint and Three Points**.

Specify the location of the ellipse centerpoint and then points **p1**, **p2** and **p3**, which belong to it.

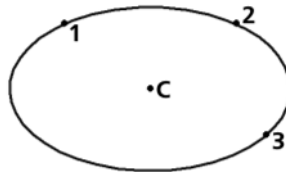


Fig. 3.2.35. Ellipse by centerpoint and 3 points

3.2.6.7. Ellipse Tangent to Two Curves



To draw an ellipse tangent to two specified objects, click **Ellipse Tangent to Two Curves**.

Specify the first and second objects at points **p1** and **p2** where they are tangent to the ellipse.



Use snaps to position the cursor precisely (see Section 3.1.1.3 on p. 870).

Specify the point **p3**, through which the created ellipse should go.

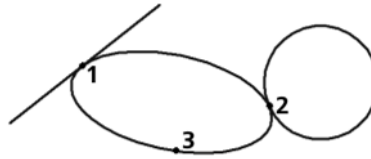


Fig. 3.2.36. Ellipse tangent to a line segment and circle

3.2.7. Arcs

This section is devoted to building circular arcs. Hereafter the word “circular” in the word combination “circular arc” will be omitted for brevity. The term “arc” should be understood to mean circular arc. Where other arc types are mentioned, e.g. ellipses, it will be stated specifically.

3.2.7.1. Arc



To draw an arbitrary arc, click **Arc**.

Set centerpoint of an arc.

Set starting point of an arc. You can do this by the following methods:

- ▼ select a point with the cursor,
- ▼ enter the angle and radius (or diameter) values in the relevant fields on the Property bar.



By default, the system awaits the radius value as you create a circle. In the **Parameter** group, the **Radius** switch is active.



To enter the diameter, activate the **Diameter** switch.

Set arc ending point. You can also do this in two ways:

- ▼ select a point with the cursor,
- ▼ enter the value of the end angle of the arc in the relevant field on the Property bar.

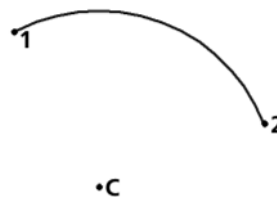


Fig. 3.2.37. Building an arc by centerpoint and two points

3.2.7.1.1. Selecting the Arc Direction



By default, the arc is built counterclockwise. In this case, the **Counter-Clockwise** switch is active in the **Direction** group on the Property bar.



To reverse the direction, activate the **Clockwise** switch.

The direction is set similarly when you build arcs of other types¹.

3.2.7.2. Arc by Three Points



To draw an arc with specified end points and a point on the arc, click the **Arc by Three Points** command.

Specify the arc start (**p1**), the point at which the arc should go (**p2**), and the arc end (**p3**) in any order. The arc center and its radius will be calculated automatically.

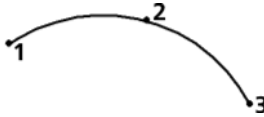


Fig. 3.2.38. Creating an arc by three points

3.2.7.3. Arc Tangent to Curve



To draw an arc tangent to a specified object, click **Arc Tangent to Curve**.

Select an object, to which the arc should be made tangent.

The **p1** point position, through which the arc should go.

Select the end point **p2** of the arc.

The arc center coordinates and its radius will be calculated automatically. Start point of arc is the tangent point.



You can enter the arc radius or diameter in the relevant field on the Property bar before specifying the point **p2**. However, building a tangent arc is possible not with all combinations of the point location on the arc (**p1**) and the radius value. If the arc ghost disappears after you enter the radius value, this means that building the arc is impossible.

The ghosts of all arc configurations satisfying the specified parameters will appear on the screen.

Activate a suitable ghost and fix it.

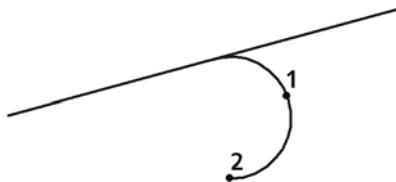


Fig. 3.2.39. Arc tangent to a line segment



To proceed to building arcs tangent to another curve, click the **Reselect** button and proceed to select a new reference curve with the pointer.

1. When you build an arc by three points, its direction is determined by the sequence in which you specified the points, which is why there will be no direction switch on the Property bar.

3.2.7.4. Arc by two points



To draw an arc with specified end points, click **Arc by 2 Points**.

Enter the radius or diameter value of the arc you are creating in the relevant field on the Property bar.



By default, the system awaits the radius value as you create a circle. In the **Parameter** group, the **Radius** switch is active.



To enter the diameter, activate the **Diameter** switch.

Specify the start point **p1** of the arc.

Select the end point **p2** of the arc.



To build an arc by diametrically opposite points, you do not have to enter the radius or diameter value; it will be determined automatically after you specify the points.

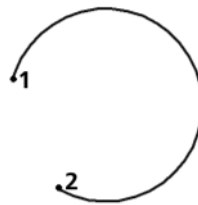


Fig. 3.2.40. Arc by two points

3.2.7.5. Arc by Two Points and Angle



To draw an arc beginning and ending at specified points and having a specific included angle, click **Arc by 2 Points and Angle**.

Enter the included angle value in the relevant field on the Property bar (the default value is 90°).

Specify the start point **p1** of the arc.

Select the end point **p2** of the arc.

The arc center coordinates and its radius will be calculated automatically.



Fig. 3.2.41. Arc by two points and angle 60°

3.2.7.6. Ellipse Arcs

KOMPAS-3D does not have a special command for building elliptical arcs. However, you can create elliptical arcs by applying trim commands (see Section 3.4.7 on p. 1137) to whole el-

lipses. Using snaps (see Section 3.1.2 on p. 871) and auxiliary elements (see Section 3.2.3 on p. 901), you can create elliptical arcs with specific parameters.

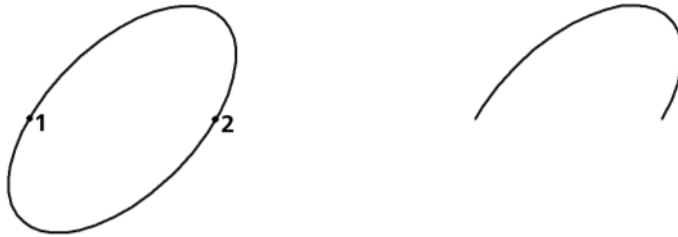


Fig. 3.2.42. Sample of elliptical arc obtained by trimming a whole ellipse at points **p1** and **p2**

Elliptical arcs are edited by means of handles (see Section 3.4.1.1.3 on p. 1114).

3.2.8. Rectangles and polygons

When you build rectangles and polygons with an even number of angles, axis lines can be drawn automatically. You can control the drawing of axis lines in the same way as when you create circles (see Section 3.2.5.1.1 on p. 910).

In KOMPAS-3D, rectangles and polygons are whole objects and not just sets of line segments. They are selected and edited wholly.

3.2.8.1. Rectangle



To draw an arbitrary rectangle, click **Rectangle**.

There are two methods of creating rectangles:

- ▼ By opposite two vertices.



Enable the **By two vertices** switch in the **Method** group of the Property bar. Specify the opposite vertices of the rectangle: **p1** and **p2**. If necessary, specify the rectangle's angle of slope in the **Angle** field. The rectangle's height and width are defined automatically.

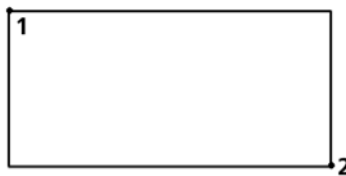


Fig. 3.2.43. Creating a rectangle by opposites vertices

- ▼ By center and vertex.



Enable the **By center and vertex** switch in the **Method** group of the Property bar. Specify the rectangle's center and one of its vertices – **p1**. If necessary, specify the rectangle's angle of slope in the **Angle** field. The rectangle's height and width are defined automatically.

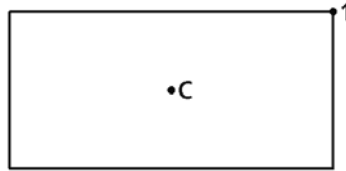


Fig. 3.2.44. Creating a rectangle by centerpoint and vertex

The angle of the rectangle's slope is counted from the X axis of the current coordinate system. By default, the angle is equal to zero.



In order to build a square, specify the second point keeping the `<Ctrl>` key pressed.

3.2.8.2. Rectangle by three points



In order to build a rectangle by three points, call the **Rectangle by three points** command.

There are two methods of creating rectangles:

▼ By three vertices.



Enable the **By three points** switch in the **Method** group of the Property bar. Specify the rectangle's vertices: **p1**, **p2** and **p3**. The rectangle's height, width and angle of slope are defined automatically.

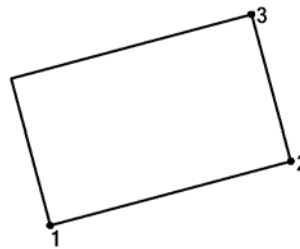


Fig. 3.2.45. Creating a rectangle by three vertices

▼ By centerpoint, midpoint of a side and vertex.



Enable the **By center and two points** switch in the **Method** group of the Property bar. Set the rectangle's center, midpoint **p1** of a side and vertex **p2**. The rectangle's height, width and angle of slope are defined automatically.

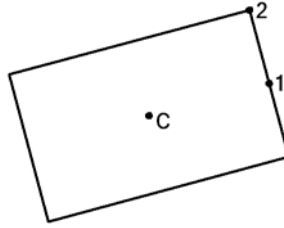


Fig. 3.2.46. Creating a rectangle by centerpoint, midpoint of a side and vertex

3.2.8.3. Polygon



To draw a regular polygon, click **Polygon**.



By default a polygon is built by an inscribed circle. The relevant switch is enabled in the **Construction method** group. To enable the construction by a circumscribed circle, activate the **By Circumscribed Circle** switch.



Enter the number of the polygon vertices in the relevant field on the Property bar.

Specify the polygon centerpoint.

- ▼ If you know the location of point **p**, i.e., of one of the vertices (when building by a circumscribed circle) or midpoint of one of the sides (when building by an inscribed circle) of the polygon you are creating, specify this point.
- ▼ If you know the polygon radius or diameter as well as the tilt ¹angle enter them in the relevant fields on the Property bar.



By default, the system awaits entry of the diameter value upon circle parameter input. In the **Parameter** group, the **Diameter** switch is active.



To enter the radius, activate the **Radius** switch.

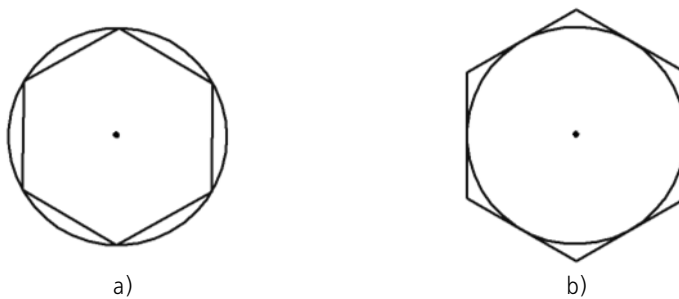


Fig. 3.2.47. Hexagon: a) by circumscribed circle, b) by inscribed circle

1. The polygon tilt angle is determined as the angle between the X-axis of the current coordinate system and the radius vector constructed from the polygon center into its first vertex (when constructing by the circumscribed circle) or into the middle of the first side (when constructing by the inscribed circle).

3.2.9. Polyline and splines

3.2.9.1. Polyline



To draw a polyline consisting of straight lines, click **Polyline**.

Specify polyline vertices.

Each vertex of the polyline is characterized by a set of parameters displayed in the **Vertex coordinates** table (see Section 3.2.9.1.1 on p. 923).

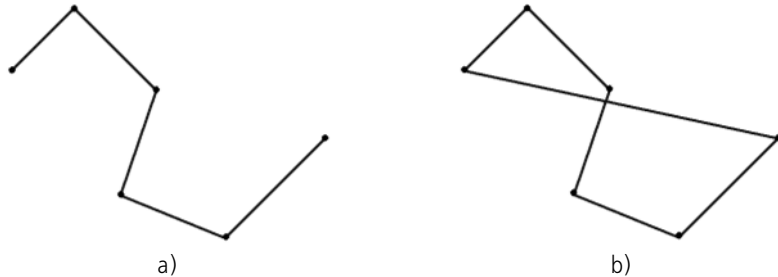


Fig. 3.2.48. Polyline: a) open, b) closed



Select an option of curve construction: closed or open. To do this, activate the desired switch in the **Mode** group: **Open object** or **Closed object**.

You can modify the polyline configuration without exiting the command. To do so, click the **Edit points** button on the Advanced Control Panel. The system will go into the object defining points editing mode. Make the required changes (see Section 3.2.9.1.2).



The created polyline is a whole drawing object. It will be selected, edited, and deleted entirely.

Fix the created polyline by clicking the **Create Object** button.

3.2.9.1.1. Table of Vertex Coordinates

Every vertex of a polyline or a spline is characterized by a set of parameters. Their values are displayed in the **Vertex Coordinates** table, which is located on the Properties Bar.

Each row of the table corresponds to a single vertex.

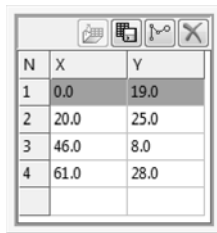
The following parameters are common for all splines and polylines:

- ▼ vertex number;
- ▼ vertex coordinates on axis X and axis Y.

When constructing a spline by points, the table displays both common parameters and ones dependent on the type of curve:

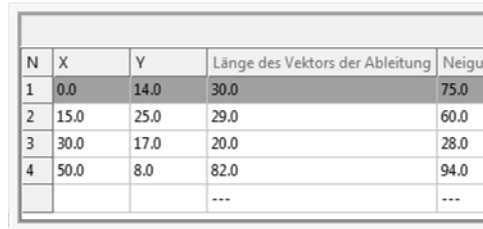
- ▼ NURBS-curve by points:
 - ▼ derivative vector length;
 - ▼ derivative vector slope angle;
 - ▼ curvature.
- ▼ Bezier Curve:

- ▼ length 1, 2 — lengths of the first and second derivative vectors;
- ▼ angle 1, 2 — slope angles of the first and second derivative vectors.



N	X	Y
1	0.0	19.0
2	20.0	25.0
3	46.0	8.0
4	61.0	28.0

a)



N	X	Y	Länge des Vektors der Ableitung	Neigung
1	0.0	14.0	30.0	75.0
2	15.0	25.0	29.0	60.0
3	30.0	17.0	20.0	28.0
4	50.0	8.0	82.0	94.0
			---	---

b)

Fig. 3.2.49. **Vertex Coordinates** table:
a) polyline and spline by poles, b) spline by points

The table rows are formed automatically during vertex construction in the document window. Editing a table row results in changed parameters of the respective vertex.

To change a vertex parameter, activate the respective cell in the parameters table by a double click and enter the new value in it. To move between cells, use arrow keys.

Above the **Vertex Coordinates** table are the following buttons used for working with parameters.



- ▼ **Read from file** — allows adding data stored in a file to the table. The button is available if the vertex parameters table is empty, i.e. right after the polyline/spline creation command is called. After the button is clicked a standard Select File dialog appears, where you need to select the desired fragment. The table of parameters will be filled in with the data from this file. The ghost of the spline constructed by the table data will appear in the document window.



- ▼ **Save to file** — allows saving the vertex parameters to a file for later use. Upon clicking this button, a standard Windows file save dialog will appear on the screen. In this dialog you need select a file name and the directory to save. By default the file shall be assigned the *txt* extension, but it can be changed if required. Coordinates can be saved to a file if at least one row in the table of parameters is filled.



- ▼ **Create new vertex** — allows creating a vertex next to an existing one. To do this, highlight a line with the number of the required vertex and click **Create New Vertex** or press the *<Insert>* key. In the document window a new vertex will appear. It will have the number of the current vertex plus one. Numbers of vertices starting from the one before which the new one was inserted will be increased by one. The newly created a vertex parameters line will be active.



- ▼ **Delete Vertex** — allows a vertex to be deleted. To do so, highlight the respective row of parameters and click **Delete Vertex** or press *<Delete>*. It is impossible to delete the last remaining row.

To perform actions with vertex parameters you can also use context menu commands of the **Vertex Coordinates** table.

3.2.9.1.2. Editing handles (when creating or editing an object)



You can change position, add or delete defining points of an object in the process of creation— after pressing the button **Edit Points** on the Advanced Control Panel.

In this mode, the defining points (handles) are displayed as small black squares in vertices, and the object is displayed as a ghost.

Move the cursor to a defining point that need to be edited. Shape of the cursor will change: it will turn into a four-sided arrow. Press the <Enter> key or click the left mouse button — the point will be selected.

You can edit object points as follows.

- ▼ **Point "dragging" with the mouse.**
- ▼ **Moving a point from the keyboard.**
For that, the point must be selected by pressing <Enter> key.
Move the cursor by the arrow keys.
- ▼ **Specify the Defining Point Coordinates.**
Specify new coordinates of the selected point on the Properties tab.
- ▼ **Adding or deleting a defining point.**
 - ▼ To add a point, click the mouse on the required segment of the object. It will be broken down into two parts by a new point in the specified location. You can move it to any position.
 - ▼ To delete a selected point, click the *Delete* button. This causes the defining point to disappear, and the object will be rebuilt according to the position of the remaining defining points.



When selecting a point position you can use the Snaps (see Section 3.1.1 on p. 865) and the Geometrical Calculator (see Section 3.1.2 on p. 871).

To exit the point editing mode and resume line constructing, release the **Edit Points** button.

3.2.9.2. Spline by poles

Spline by poles — a spline that goes through the extreme vertices tangent to outermost segments of the reference polyline and does not go through intermediate vertices (they are called the spine's "poles").



To construct a spline by poles, call the **Spline by poles** command.

Set spline vertices. To do so, specify the vertices of the reference polyline.

Each vertex of the spline is characterized by a set of parameters displayed in the **Vertex coordinates** table (see Section 3.2.9.1.1 on p. 923).

Set spline characteristics: spline order and weight of each handle. To do so, enter the required values in the fields **Order** and **Weight** on the Properties Bar. The order ranges from 2—10 whereas the weight falls between 0.0001—999.

For details on spline order and point weight refer to Section 2.5.1.4 on p. 329.

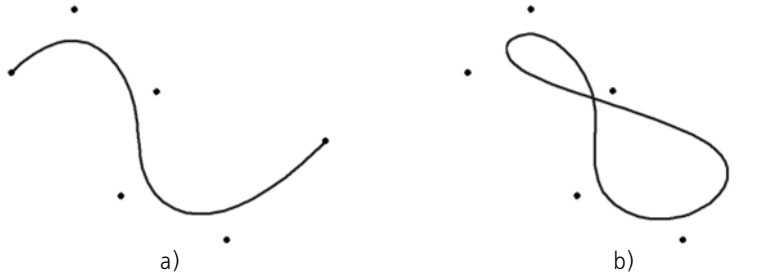


Fig. 3.2.50. Pole spline: a) open, b) closed



Select an option of spline construction: closed or open. To do this, activate the desired switch in the **Mode** group: **Open object** or **Closed object**.

You can change spline configuration without leaving the command dialog. To do so, click the **Edit points** button on the Advanced Control Panel. The system will go into the object defining points editing mode. Make the required changes (see Section 3.2.9.1.2 on p. 924).

Fix the created spline by clicking **Create Object**.

3.2.9.3. Spline by points



To construct a spline passing through given vertices call the command **Spline by points**.

Spline by points can be a curve of one of the following types:

▼ **Bezier Curve**

▼ **NURBS-point curve**

Select the required type of curve from the list **Curve Type** on the Properties Bar.

When constructing a NURBS-point curve select from the list **Method** on the Properties Bar the required method of forming parameters of spline points (see Section 3.2.9.3.2 on p. 928).

Set spline vertices. To do so, specify the points through which the spline will pass.

Each vertex of the spline is characterized by a set of parameters displayed in the **Vertex coordinates** table (see Section 3.2.9.1.1 on p. 923).



Fig. 3.2.51. Bezier Curve: a) open, b) closed



Select an option of spline construction: closed or open. To do this, activate the desired switch in the **Mode** group: **Open object** or **Closed object**.



To change spline configuration click the button **Edit Points** on the Advanced Control Panel. The system will go into the object defining points editing mode. In this mode you can:

- ▼ change the position and the number of vertices, see Section 3.2.9.1.2 on p. 924;
- ▼ change spline shape in a vertex:
 - ▼ Bezier curve — by controlling the length and direction of a derivative vector,
 - ▼ for a NURBS-point curve — by controlling tangency and curvature, see Section 3.2.9.3.1 on p. 927.



Fix the created spline by clicking **Create Object**.



All objects on drawings of the **Polyline and Splines** chapter have the same set of base points. You will notice the different shapes of objects having the same base points, depending on the object type and whether it is open or closed.

3.2.9.3.1. Controlling spline shape

When constructing a spline that is a NURBS-point curve, you can change its shape by controlling tangency and curvature in any vertex.

Tangency indicates how close is the spline's form to the tangent circle near its current vertex, curvature characterizes the radius of the circle (see Section 2.5.1.6 on p. 331 for more information about curvature and tangent circle).



Changing the spline's shape is possible in the object defining point editing mode for the current vertex. To enter the mode click **Edit Points** on the Advanced Control Panel.

After the mode is activated, the Properties Bar will have the list **Management** with spline management options in the current vertex.

Specify the vertex in which the spline shape needs to be changed in the document window or the **Vertex Coordinates** table.



Select the required option on the **Management** list:

- ▼ **Tangency** — allows setting the length and slope angle of the derivative vector in the selected vertex,
- ▼ **Tangency and curvature** — allows setting curvature, apart from the length and slope angle of the derivative vector.

After selecting one of management options, a notation of the option and managing ghost will appear next to the current vertex: one or two dashed arrows.

Set the parameters of the selected option using the Properties Bar or the managing ghost in the document window. Note that derivative vector length can only be set on the Properties Bar.

While in the spline shape management mode, you can also change the position of the current vertex by dragging it in the document window (see Section 3.2.9.1.2 on p. 924) or entering destination vertex coordinates in the parameters table (see Section 3.2.9.1.1 on p. 923).

After finishing editing the spline's form at one of vertices, you can:

- ▼ move to managing spline shape in another vertex — to do so, select the required vertex in the document window or in the table of parameters – the row with the number of that vertex,
- ▼ exit spline shape management mode — click **Edit Points** on the Advanced Control Panel,
- ▼ finish spline creation — click **Create Object**.



3.2.9.3.2. Methods of forming spline points

When constructing a spline that is a NURBS-point curve, you can select a desired curve parameterization method.

NURBS-point curve has parametric representation (see Section 2.5.1.2 on p. 327). The points used for creating the spline correspond to certain values of the parameter t . The shape of a spline passing the same points can be different (see Fig. 3.2.52). This depends on the method of forming spline point parameters:

- ▼ **Linear** — the values of the parameter that correspond to the user defined points are considered evenly distributed within the curve parametric range.
- ▼ **Chord length** — location of parameter values that correspond to user-defined points within the curve parametric range depends on the distance between neighboring points.
- ▼ **Centripetal** — location of parameter values that correspond to user-defined points within the curve parametric range depends on the square root of the distance between neighboring points.



Fig. 3.2.52. Splines with different methods of forming point parameters:
I – **Linear**, II – **Centripetal**, III – **Chord length**

To set a method of forming spline point parameters, select the required option on the **Method** list on Properties Bar.

3.2.10. Continuous Drawing of Objects



To draw a sequence of lines, arcs, and splines, click **Continuous Drawing of Objects**.

During the line input, an end point of the created object automatically becomes the start point for the next object. This command is convenient to use, for example, when creating a part contour consisting of objects of different types.

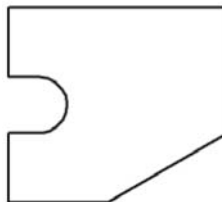


Fig. 3.2.53. Sample of sequence of objects created using the continuous drawing command











The created sequence is not a single object. Line segments will be selected, edited and deleted separately.

3.2.10.1. Creating Sequence Objects

After you click the command on the Property bar, the **Type** group will be displayed in addition to parameter entry fields. It contains switches that let you specify the type of a segment you are creating. To select a type, activate the relevant switch (see Table 3.2.2).

Tab. 3.2.2. Segment type switches

	Segment Type	Segment creation method
	Line	Arbitrary
		Parallel to an object
		Perpendicular to an object
		Tangent to an object
	Arc	By three Points
		Tangent
	Spline	Bezier Curve
		NURBS

By default, upon the first call of the command, the current type of object is **Line Segment**. This means that when you specify these points, the system will build a polyline consisting of lines that connect these points.

You can modify the current object type at any time while constructing the line. To do this, activate the desired speech in the **Type** group.

Ways to build various objects in the continuous drawing mode, as well as the techniques of controlling their parameters, are identical to the methods and techniques mentioned above for building individual objects.

3.2.10.2. Completing the Entry of Objects

The entry of objects can be completed by means of **Close** and **Draw New** switches on the Property bar.



Upon calling the **Close** command, a point that coincides with the first point of the sequence of objects is automatically created, and the sequence creation terminates. The automatically added point belongs to the object type, whose creation was active at the time of closing. If the number of already entered object points is not enough to automatically create an object to close the sequence, then the **Close** switch is inaccessible. After closing up the input sequence, the system waits for the input of a new continuous sequence of objects.



After activation of the **Draw New** switch, the drawing of a sequence is completed without closing and the system awaits entry of a new sequence.

3.2.10.3. Line Style during Continuous Drawing of Objects

The current style applies only to the object being formed and not to all objects in the sequence. This is why you can draw objects of different styles without interrupting the entry of the sequence.

3.2.11. Line



To draw a line consisting of line segments and/or circular arcs whose position can be either determined automatically by the system depending on the selected drawing objects, or specified by the user, click **Line**.

During the line input, an end point of the created object automatically becomes the start point for the next object.



The created sequence is not a single object. Arcs and line segments will be selected, edited, or deleted separately.

Two object types can be used to compose a line: a straight segments and an arc. You can modify the current object type at any time while constructing the line.

The method of constructing a straight segment or arc is determined by the system automatically depending on what objects are specified by the user. For example, if a straight object is specified when constructing a line segment, then the segment can be parallel or perpendicular to it. This is determined by the cursor motion direction.

When creating the first object in a sequence, there are additional construction methods, since this object, unlike the following ones, has none of its points fixed.

Ways to create the first objects of the line are described in Sections 3.2.11.2 on p. 932 and 3.2.11.4 on p. 936, and ways to create subsequent objects—in Sections 3.2.11.3 on p. 935 and 3.2.11.5 on p. 937.



The **Draw New** switch on the Property bar serves to terminate current line creation and start a new one.



Continuous Drawing of Objects switch lets you specify whether you will be creating individual objects or their sequence. By default, when the **Line** command is called for the first time, this switch is active and the start point of each subsequent object coincides with the end point of the previous one. To be able to define start points of objects arbitrary, disable the **Continuous Drawing of Objects** switch. To return to the chain creation, activate the switch again.

3.2.11.1. General Procedure of Building Line Objects

1. Specify the segment type by activating the relevant switch in the **Type** group on the Property bar:



▼ **Line segment,**



▼ **Arc.**



The current segment type of a line can be changed without referring to the Property bar by using the keys **<Ctrl> + <Tab>**.

2. Specify an object(s) for building a segment. For example, a line segment or arc can be tangent to a circle, a line segment can be parallel or perpendicular to a straight line or be located on an angle bisector, etc.

To select an object, move the cursor to it. When the object gets highlighted, left-click.



If you cannot select an object, verify that the **Point on Curve** snap is disabled. If the snap is enabled, disable it.

The failure to select an object when the snap is active is because the latter has a higher priority than the object selection.



To change the object, click the **Reselect** button on the Advanced Control panel. Highlighting will be cleared off the previously selected object, and you can select a new object.

After you select an object(s), a ghost will appear on the screen: a straight line or arc containing a segment.

3. Specify the start and (or) end points of the object. If the start (end) point is specified outside of the segment ghost, the location of the object start (end) is determined by projection of the specified point on the ghost.

While constructing segments of a line, the object parameter fields are displayed on the Property bar: the start and end point coordinates, the radius or diameter (for arcs), the length and angle (for line segments). These fields are used in the same way as when creating segments and arcs using the dedicated commands.

Ways of building line objects are detailed in chapters 3.2.11.2–3.2.11.5. In some cases, the sequence of actions may differ from the general procedure mentioned above.

After you create the object, the system waits to build the next object of the same type.

Note the following specifics of building line segments. In the cases when there are several possible methods to construct segments, the one is selected that is nearest to the selection point of the object. This is the difference between creating line segments and their counterparts created by dedicated commands. This approach helps speed up the construction, but requires a certain skill.

As we know, there are two variants of a line going through a given point and tangent to a circle. When constructing a line segment tangent to an arc by the **Line Tangent through Outside Point** command, you can select one of the tangency variants; you do not have a choice, however, when constructing a tangent segment as part of a line. The segment will be positioned on the line that is tangent to the circle at the point nearest to the circle selection point.

3.2.11.2. Ways to Build the First Line Segment

3.2.11.2.1. Parallel or Perpendicular to Object

To draw the first line segment parallel or perpendicular to another object, complete the following steps.

1. Use the cursor to select a straight object to which the line segment being created should be made perpendicular or parallel.
2. Set the start point of the segment.

The order of selecting a straight object and the start point of the segment doesn't matter.

3. Move the pointer across the document area. The line segment ghost will be displayed. Depending on the cursor position relative to the start point, the ghost may be parallel or perpendicular to the object. If the cursor is in area 1 (Fig.3.2.54, a), the line ghost is built parallel to the reference object (Fig. 3.2.54, b), and if it is in area 2, perpendicular to the reference object (Fig. 3.2.54, c).

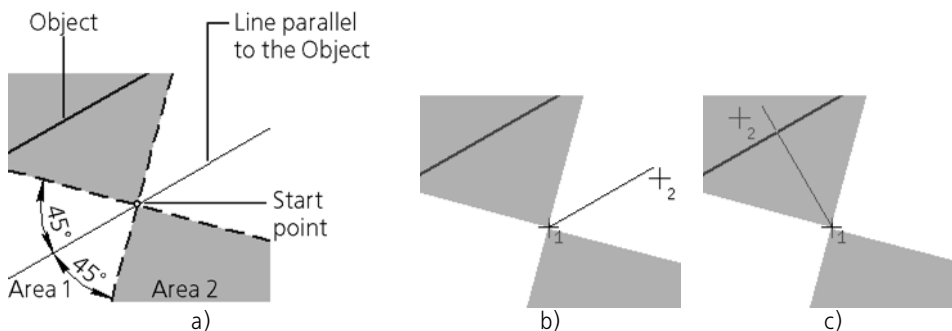


Fig. 3.2.54. Building a line parallel/perpendicular to a straight object

4. Move the cursor to set the desired segment direction and specify its end point.

3.2.11.2.2. Tangent to Object

To draw the first line segment tangent to an object, complete the following steps.

1. Use the cursor to select an curvilinear object, to which the line segment being created should be made tangent.
2. Move the pointer across the document area. You will see how the tangent line ghost follows the cursor.

To change the direction of the tangent to the circle (ellipse), move the cursor so as to cross the circle (ellipse).

3. Set the start and end point of the segment.

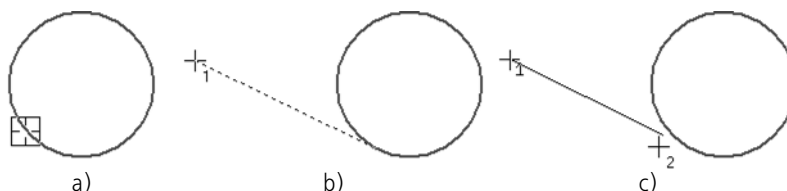


Fig. 3.2.55. Constructing of the first line segment tangent to another object:
a) selecting an object, b) selecting the segment start point, c) selecting the segment end point



To start (finish) the segment exactly at the tangency point, use the **Tangency** snap (see Fig. 3.2.59).

3.2.11.2.3. Tangent to One Object and Parallel to Another

To draw the first line segment tangent to an object and parallel to another one, complete the following steps.

1. Use the cursor to select an curvilinear object, to which the line segment being created should be made tangent.

Similarly, specify a straight object to which the second should be made parallel.

The sequence of selecting objects is not relevant.

2. Move the pointer across the document area. You will see how the tangent line ghost follows the cursor.
3. Set the start and end point of the segment.

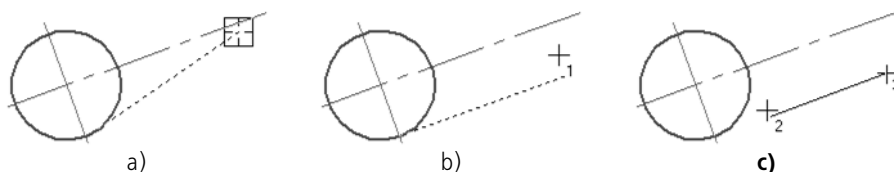


Fig. 3.2.56. Constructing of the first line segment tangent to one object and parallel to another: a) specifying objects, b) specifying the start point of a line, c) specifying the end point of a line



To start (finish) the segment exactly at the tangency point, use the **Tangency** snap (see Fig. 3.2.59).

3.2.11.2.4. In between Two Parallel Lines

To draw the first line segment positioned on a line going in between two straight objects parallel to each other, complete the following steps.

1. Select the first and second straight object with the cursor.
2. Set the start point of the segment.
3. Move the pointer across the document area. A ghost segment will be displayed, positioned on a line centered between two selected straight objects.
4. Set the end point of the segment.

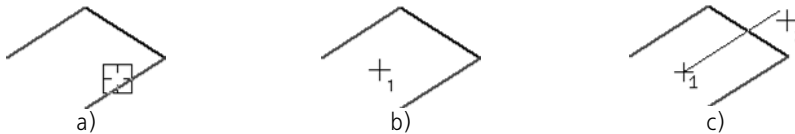


Fig. 3.2.57. Constructing the first line segment on a line centered between two parallel lines: a) selecting objects, c) selecting the segment end point

3.2.11.2.5. Tangent to Two Objects

To draw the first line segment tangent to two objects, complete the following steps.

1. Select the first and second curvilinear objects with the cursor, to which the segment being created should be made tangent.
- After selecting the second object, a ghost segment will be displayed, positioned on a line tangent to the selected objects at the points nearest to those, at which the objects were picked.
2. Set the start and end point of the segment.

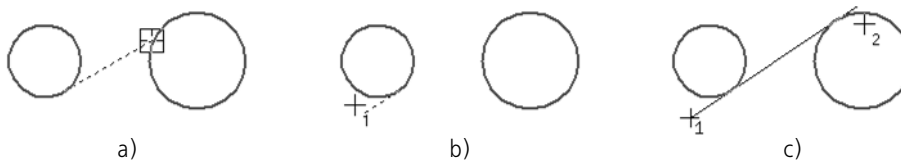


Fig. 3.2.58. Constructing the first line segment tangent to two objects: a) selecting objects, b) selecting a segment start point, c) selecting the segment end point



To start (finish) the segment exactly at the tangency point, use the **Tangency** snap (see Fig. 3.2.59).



Fig. 3.2.59. Example of constructing the first line segment

3.2.11.2.6. On the Angle Bisector

To draw the first line segment positioned on the bisector of the angle between two straight objects, complete the following steps.

1. Select two straight nonparallel items with the mouse cursor.
2. Set the start point of the segment.

Move the pointer across the document area. A ghost segment will be displayed positioned on the bisector of the selected angle.

3. Set the end point of the segment.

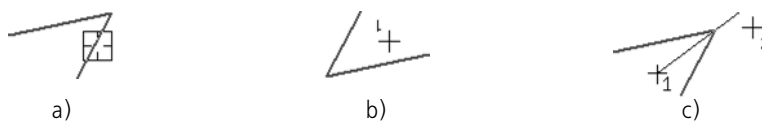


Fig. 3.2.60. Constructing the first line segment positioned on an angle bisector:
 a) specifying objects, b) specifying the start point of a line,
 c) specifying the end point of a line

3.2.11.3. Ways to Build Subsequent Line Segments

3.2.11.3.1. Parallel or Perpendicular to Object

To draw a subsequent line segment parallel or perpendicular to another object, complete the following steps.

1. Use the cursor to select a straight object to which the line segment being created should be made perpendicular or parallel.
2. Move the pointer across the document area. The line segment ghost will be displayed. Depending on the cursor position relative to the start point, the ghost may be parallel or perpendicular to the object Fig.3.2.54 on p. 932).
3. Move the cursor to set the desired segment direction and specify its end point.

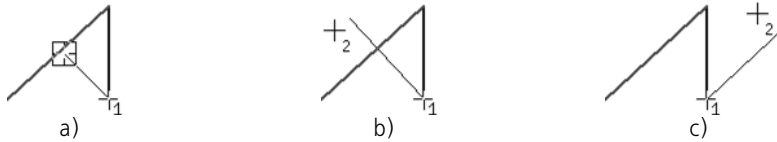


Fig. 3.2.61. Constructing a subsequent line segment perpendicular/parallel to another object: a) specifying an object, b) drawing a perpendicular line, c) drawing a parallel line

3.2.11.3.2. Tangent to Object

To draw a subsequent line segment tangent to an object, complete the following steps.

1. Use the cursor to select an curvilinear object, to which the line segment being created should be made tangent.
2. Move the pointer across the document area. You will see how the tangent line ghost follows the cursor. It is positioned on a line tangent to the selected object at the point nearest to the one at which that object was picked.
3. Set the end point of the segment.



Fig. 3.2.62. Constructing a subsequent line segment tangent to another object: a) selecting the object, b) selecting the segment endpoint



To finish the segment exactly at the tangency point, use the **Tangency** snap (see figures. 3.2.63).



Fig. 3.2.63. Example of constructing a subsequent line segment

3.2.11.4. Ways to Build the First Arc

3.2.11.4.1. Tangent to Object

To draw the first arc of the line tangent to an object, complete the following steps.

1. Set the start point of an arc.
2. Select an object, to which the arc should be made tangent.

3. Move the pointer across the document area. You will see how the arc ghost moves along. If you need to flip the arc direction, move the cursor through its start point.

The ghost is made tangent to the selected object at the point in which that object was selected. The arc radius is determined automatically and fixed.

 - ▼ To modify the radius, enter the desired value in the **Radius** or **Diameter** field on the Property bar.
 - ▼ To define the radius by specifying the end point, unfix the **Radius** or **Diameter** field, but do not enter a value.
4. Set the end point of an arc.

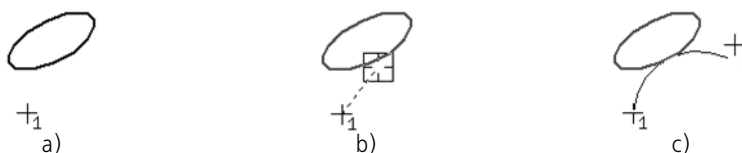


Fig. 3.2.64. Constructing the first arc of a line tangent to an object:
a) selecting the start point, b) selecting the object, c) selecting the end point

If the positions of the start and end point of the arc are fully constrained, then those uniquely define the arc radius. In this case, you can first specify the start and end point of arc, and then the object to which it should be tangent (Fig.3.2.65). The arc will be fixed with an automatically determined radius.



Fig. 3.2.65. Constructing the first arc of a line tangent to an object:
a) selecting the start and end point, b) selecting the object

3.2.11.4.2. Arbitrary

To draw the first arc arbitrarily, complete the following steps.

1. Set the start and end point of the arc. The arc ghost will be displayed.
2. Enter from the keyboard the arc radius or select with the mouse the point through which the arc should go.

3.2.11.5. Ways to Build Subsequent Arcs

3.2.11.5.1. Tangent to Object

A subsequent line arc can be drawn tangent to:

- ▼ previous line segment,
- ▼ previous segment and an arbitrary object,

▼ arbitrary object.

By default, the next arc in the line is constructed tangent to the previous segment. The segment is highlighted in the document window.

To build an arc tangent to a previous segment, perform the following steps.

1. If you know the arc radius or diameter, enter the required value in the **Radius** or **Diameter** field on the Property bar and fix it.
2. Move the pointer across the document area. You will see how the arc ghost moves along. If you need to flip the arc direction, move the cursor through its start point.
3. Set the end point of an arc.

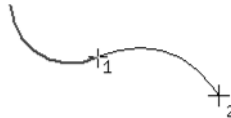


Fig. 3.2.66. Building a subsequent arc tangent to a previous line segment

To build an arc tangent to a previous segment and an arbitrary object, perform the following steps.

1. Select the second object, to which the arc should be made tangent. It will be highlighted in the document window.
2. Move the pointer across the document area. You will see how the arc ghost moves along. The arc radius cannot be modified— it is fully constrained by the two objects to which the arc is tangent. If you need to flip the arc direction, move the cursor through its start point.
3. Set the end point of an arc.



Fig. 3.2.67. Constructing a subsequent arc of a line tangent to the previous segment and to an arbitrary object: a) object selection, b) specifying the arc endpoint

To build an arc tangent to an arbitrary object, perform the following steps.



1. Click the **Reselect** button on the Advanced Control panel. Highlighting will be cleared from the previous segment.
2. Select an object, to which the arc should be made tangent.
3. Move the pointer across the document area. You will see how the arc ghost moves along. It is made tangent to the selected object at the point in which that object was selected. The arc radius is determined automatically and fixed. To change the radius or diameter, enter the required value in the **Radius** or **Diameter** field on the Property bar. The ghost will rebuild.

If you need to flip the arc direction, move the cursor through its start point.

4. Set the end point of an arc.



Fig. 3.2.68. Constructing a subsequent line arc tangent to an arbitrary object: a) object selection, b) specifying the arc endpoint

3.2.11.5.2. Normal to the Previous Segment

A subsequent arc of a line can be drawn tangent to the normal to the previous line segment constructed at its end point. The tangency point is the start point of the arc. Construction of an arc normal to the previous segment is shown on Fig.3.2.69. The previous segment is drawn in a normal line, the dashed line is the normal to it at its end point, and the thin line is the arc along the normal.

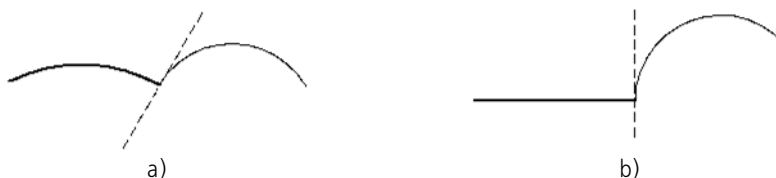


Fig. 3.2.69. Arc by normal: a) to an arc, b) to a line segment

To proceed To drawing a normal arc, press the <Ctrl> key, move the mouse and release the key. You will see that the ghost arc rebuilds so as to be tangent to the previous segment normal constructed in its end point.

An arc by a normal can be tangent to an object or be arbitrary.

To draw an arc normal tangent to an object, complete the following steps.

1. Select an object, to which the arc should be made tangent. To do this, move the cursor to the desired object. When the object gets highlighted, left-click.
2. Move the pointer across the document area. You will see how the arc ghost moves along. The arc radius cannot be modified—it is fully constrained by the two objects to which the arc is tangent.

If you need to flip the arc direction, move the cursor through its start point.



If you need to change the object to which the arc is tangent, click the **Reselect** button on the Advanced Control panel. Highlighting will be cleared off the previously selected object, and you can select a new object.

3. Set the end point of an arc.



Fig. 3.2.70. Arc by a normal, tangent to an object:
a) object selection, b) specifying the arc endpoint

To draw an arbitrary normal arc, complete the following steps.

1. If you know the arc radius or diameter, enter the required value in the **Radius** or **Diameter** field on the Property bar and fix it.
2. Move the pointer across the document area. You will see how the arc ghost moves along.
If you need to flip the arc direction, move the cursor through its start point.
3. Set the end point of an arc.

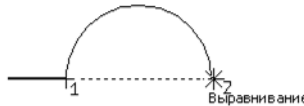


Fig. 3.2.71. Constructing an arbitrary arc by a normal

A consequent arc of a line will also be constructed by a normal. To go back to drawing tangent arcs, press the <Ctrl> key, move the mouse and release the key. The arc ghost will rebuild.

3.2.11.5.3. Arbitrary

By default, the next arc in the line is constructed tangent to the previous segment. The segment is highlighted in the document window.



To cancel building a tangent arc, click the **Reselect** button on the Advanced Control panel. Highlighting will be cleared from the previous segment. Then, complete the following actions.

1. Set the end point of an arc. The arc ghost will be displayed.
2. Type in the arc radius or diameter in the **Radius** or **Diameter** field or click a point through which the arc should pass.

3.2.12. Multiline

3.2.12.1. General information

Multiline is a geometric object which consists of one or several lines built equidistant from the base line. Multiline may be used to draw piping, walls, partitions, fences, and other extended objects with a contour made of several lines (Fig. 3.2.72).

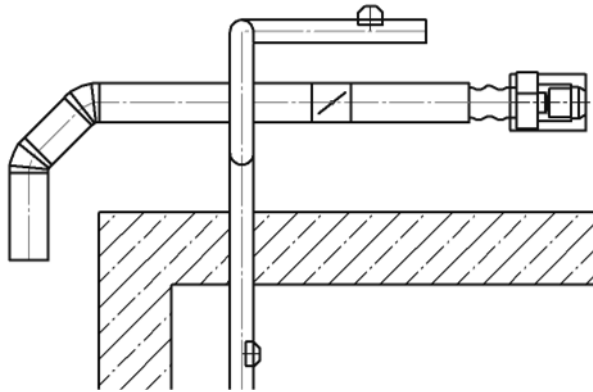


Fig. 3.2.72. Example of using multilines



It is not recommended to represent several objects with a single multiline (e.g. the group of parallel pipelines).

Lines in a multiline may have various styles.

The maximum number of lines in a multiline is 64.

All lines of a multiline are laid in one layer.

It is possible to change a configuration of a base line.

You can configure default multiline parameters in the setup dialog (see Section 9.2.6.9 on p. 1770) and also customize user parameters during multiline creation.

The multiline parameters can be saved in the multiline template file for further use. Using them you can create multilines of the same type.

Multiline may be included into a macroelement and a group.

3.2.12.2. Terms and Definitions

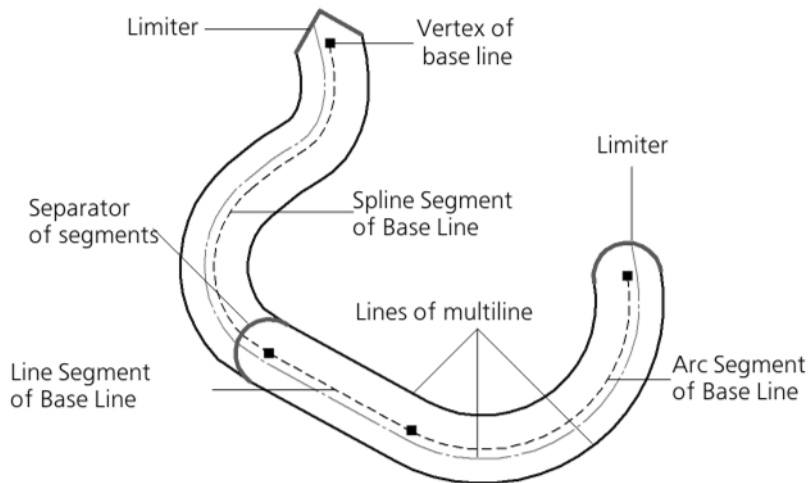


Fig. 3.2.73. Multiline (base line is shown as a dashed line)

Base line of a multiline— an invisible line of a multiline, the vertices of which are defined during the creation of a multiline and from which all other lines of the multiline are equidistantly built.

Segment of the base line of a multiline— a part of the base line between the neighboring vertices.

Line of a multiline— a complex geometric object consisting of successively connected linear, spline segments, circular, and elliptical arcs.

Segment of a multiline— a part of a multiline built based on one segment of the base line.

Multiline limiter— a line connecting end points of the outermost lines of the end multiline segment in a certain manner.

Separator of multiline segments— a line connecting the outermost lines of a multiline at the segment joint.

The **multiline's width** is the distance between the outmost lines of the multiline; the base line is not considered an outmost one. The width of a multiline consisting of a single line is equal to 0.

Multiline type— a set of the following multiline settings:

- ▼ Number of multilines and their settings— an offset from the base line and style;
- ▼ type of multiline limiters and their settings— height and style of a line

Same-type multilines— these are the multilines of one type. But the location of the base line vertices, their number, and separators of segments in the same-type lines may be different.

3.2.12.3. Creation of a multiline



To create a new multiline, call the **Multiline** command. The controls of multiline construction will appear on the Properties toolbar (Fig.3.2.74).

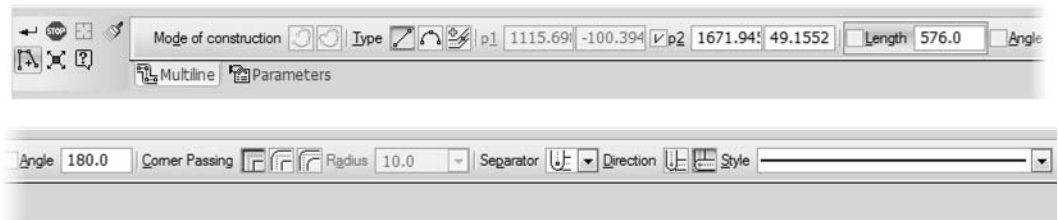


Fig. 3.2.74. Property toolbar for multiline construction



By default adding of segments to a multiline is switched on— **Add segment** button is depressed.

Specify the multiline parameters: number, offset and styles of lines. This can be completed by the following methods:

- ▼ define settings on the Property toolbar (See Section 3.2.12.4 on p. 943),
- ▼ get the settings from the multiline template (See Section 3.2.12.5 on p. 945),
- ▼ copy the settings of a previously built multiline (See Section 3.2.12.6 on p. 946).

Add the segments of the base line (see Section 3.2.12.6 on p. 946) and select the methods of passing the corner in vertices — variants of the multiline lines connection on segment joints (see Section 3.2.12.7 on p. 948).

The document window displays the ghost of the multiline being created. You can edit base line configuration directly while creating the multiline (see Section 3.2.12.10 on p. 952)— change vertex position and corner rounding method in it as well as add or delete a vertex. All changes made have an immediate effect on the multiline ghost.



Construction mode group of switches determines whether it's necessary to close a multiline. The multiline is closed by a straight segment.



A multiline comprising a single straight or spline segment cannot be closed.

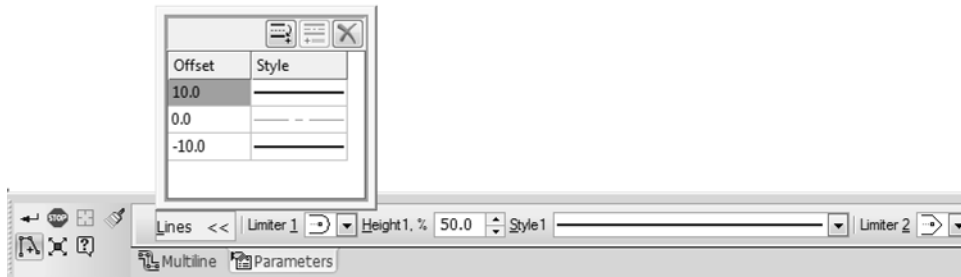


For open multiline you may define a type of limiters (See Section 3.2.12.8.1 on p. 951) and their settings: height and style of drawing lines (See Section 3.2.12.8.2 on p. 951).

To complete the creation of a multiline, click the **Create Object** button on the Special Control Panel.

3.2.12.4. Lines of a multiline

Multiline lines are listed on the **Lines** of the **Settings** tab (Fig. 3.2.75).

Fig. 3.2.75. Lines **Toolbar**



The styles of lines and their offset in relation to the base line are displayed in the list. The lines are listed in the ascending order of their offsets. Offset limits: from -50000000 to 50000000. The default line set is specified during the setup of multiline settings for new or current drawings (See Section 9.2.6.9 on p. 1770).

The **Lines** toolbar also contains buttons to control the lines. The description of the buttons is given in Table 3.2.3.

Tab. 3.2.3. Multiline lines control

The	Description
	<p>Create a new line Adds a new line to a multiline.</p> <p>After pressing the key the multiline is supplemented with a new line, and a new string containing the new line parameters is added to the list. The line has <i>Normal</i> style by default and a negative offset. Adding step is <i>10</i>. You may also add a new line by pressing the <i><Ins></i> key.</p> <p>To change an offset, enter the required value in the first field of the line, to change the style — select the required variant from the list in the second field.</p>
	<p>Create a symmetrical line Adds a line to the multiline which is symmetrical to the line highlighted in the list.</p> <p>After pressing the key the multiline is supplemented with a new line, and a new string containing the new line parameters is added to the list. The added line has the style of the highlighted line and its offset with an opposite sign. The new line on the list has a position corresponding to its offset.</p> <p>The button is unavailable if in the list there's selected the line that already has a symmetrical one or if the selected line has zero offset.</p>

Tab. 3.2.3. Multiline lines control

	The	Description
	Delete line	<p>Deletes the line highlighted in the list from the multiline. After pressing the key the highlighted line will be deleted from the multiline and from the list.</p> <p>You can also delete the line by pressing the <code><Ctrl>+<Delete></code> key combination.</p> <p>The key is not available if there is just one line in the list or if all lines are highlighted.</p>
		<p>Degenerate lines of a multiline are not displayed in a document. If such lines appear in a multiline a message flashes on the screen to signal that it will be impossible to display them. Degenerative lines are not removed from the multiline— they are seen in the list of multilines on the Lines toolbar of the Settings tab. If the form of the baseline and/or values of offsets are changed, these lines become visible.</p> <p>If all lines become degenerative during the multiline construction, only the ghost of the basic line will be displayed, and the Create object button will become unavailable.</p>

3.2.12.5. Multiline template

The multiline parameters can be saved in the multiline template file for further use. A template file has *mlt* extension.

Templates of multilines are located in the folder defined by the KOMPAS-3D environment variable MULTILINETEMPLATES. By default, this folder is *\Templates* in the main system installation folder. In order to define another folder for multiline templates, create and edit the *KOMPAS.ini* file (for more information about the KOMPAS-3D environment's variables, see Section 9.5.4 on p. 1886; for more information about *KOMPAS.ini* file see Section 9.5.3).

The multiline's template contains the parameters of the multiline:

- ▼ the number of lines and their parameters: style and displacement from the base line;
- ▼ the types of limiters on the multiline ends and their parameters: the limiters height and the lines' style.



To save settings of the current multiline, press the **Save template** button in the **Settings** tab of the Properties toolbar. Windows-standard File Save dialog will be displayed. Enter a template name and click the **Save** button.



To use the template in creation and editing of a multiline, press the **Load Template** button in the Settings tab of the Properties toolbar. The standard File Open dialog will appear on the screen. Select a file name and click the **Open** button. The multiline will receive parameters from the template. The multiline's ghost will be changed in line with the obtained parameters.

If required, you may edit the settings received from the multiline template (See Section 3.2.12.4 on p. 943 and Section 3.2.12.8 on p. 951).



If the message *Cannot open file* pops up during the template loading, this means that the selected file is not a template or the template file is incorrect.

3.2.12.6. Base line



The base line of a multiline consists of segments.



The group of switches **Type** (Fig.3.2.76) allows to select type of the segment to be added to the base line:



Straight-Line Segment or **Arc Segment** (see Section 3.2.12.6.1 on p. 946);

Segment by Object (see Section 3.2.12.6.2 on p. 946).

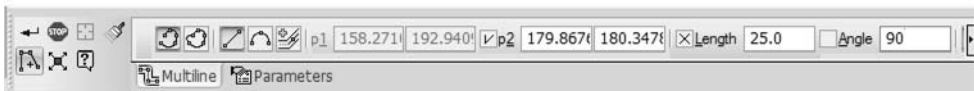


Fig. 3.2.76. The group of switches **Type**

You can add segments to the base line when creating and editing a multiline.

3.2.12.6.1. Straight-line and arc segments



Straight-line and arc segments can be added to the base line by constructing them directly during the creation (or editing) of the multiline.



To add a straight-line or arc segment, enable the required switch in the **Type** group: **Straight-line Segment** or **Arc Segment**. Build a segment of the base line. The procedure for construction of the straight-line and arc segments is similar to that for the line objects (See Section 3.2.11.1 on p. 931).

Please, pay attention to the following features of construction of base line segments.

- ▼ By default an arc segment is made on the tangent to the previous segment of any type.
- ▼ By default the straight-line segment is made on the tangent to the previous arc segment.



To cancel the construction of the segment on the tangent to the previous segment, press the **Define Again** key on the Special Control Panel. You may then randomly construct an oriented segment of the base line.

The segment adding is reflected in the ghost of the multiline.

After the segment adding the system will wait for the following segment of the same type. The end vertex of the created segment will automatically become a start vertex of the following segment.

3.2.12.6.2. By Object Segment

The opened geometrical object previously created in the graphical document may be added to the base line.

- ▼ segment;

- ▼ arc of a circle or ellipse;
- ▼ NURBS, Bezier curve or polyline;
- ▼ contour;
- ▼ multiline.



To add the opened geometrical object to the base line, activate the **Segment by Object** switch in the **Type** group.

Define an object to add to the base line. To do this, move the cursor to it. When the object gets highlighted, left-click.



If the multiline already has segments, the object adding is possible only if the end vertex of the last segment matches the start or end point of the added object.

Several successively connected objects may be added to the base line. These objects should be specified by their location order.

The Bezier curve, NURBS and the polyline should be added to the base line in whole. The Bezier curve or NURBS is added as one segment, the polylines added as several segments the number of which is equal to the number of the polyline segments.

You may define another multiline as an object. In this case the base line will be added with all segments of the defined multiline with its specified methods of passing the corner in the base line vertex (See Section 3.2.12.7 on p. 948).



To cancel the object selection, press the **Cancel** button on the **Standard** bar.



Control Mode group switches specify whether to leave initial objects after they've been added to the base line.



3.2.12.6.3. Base line modification

After the multiline is built, its base line may be added with straight-line or arc objects using the **Chamfer on Object Corners** (See Section 3.2.15.2 on p. 972) and **Fillet on Object Corners** (See Section 3.2.15.4 on p. 973). Vertices of new segments will have default settings — the corner passing with cut without a segment separator (See corner passing methods in Section 3.2.12.7 on p. 948).

Besides, you can add a segment at the beginning and/or in the middle of the base line of the multiline by splitting this multiline into parts and creating a new one from a new segment (segments) and parts of the original multiline.

For instance, the document has a multiline built with vertices A, B, C, D (Fig. 3.2.77, a). Add curves at the beginning and in the middle (between vertices B and C) of this multiline (Fig. 3.2.77, b).



Fig. 3.2.77. Adding segments to the multiline:
a) initial multiline; b) result of addition

For this, do the following.



1. Divide the multiline into three sections in vertices B and C. For this, use the **Divide curve** command.
2. Remove the BC section.
3. Build a curve of the required form.



The end point of the curve should match the start point of A multiline.

4. Build the second curve.



The start and end points of the curve should match B and C vertices.



5. Call the **Multiline** command.
6. Copy the initial multiline properties to the new multiline.
 - 6.1. Press the **Copy Properties** button on the Special Control Panel.
 - 6.2. Specify the properties source in the document window— any of the parts of the initial multiline.
7. Add built objects and parts of the initial multiline to the base line of the new multiline.



- 7.1. In the **Type** group enable the **Segment by Object** switch.
- 7.2. Specify objects to be added to the base line: built objects and parts of the initial multiline. These objects should be specified by their location order.



To remove the initial objects during the construction, enable the **Remove Initial Objects** switch in the **Construction Mode** group of **Settings** tab.

3.2.12.7. Methods of passing the corner in a vertex

The method of passing the corner in the base line vertex— is a method of connection of the multiline lines in the segment joint. There are three methods of corner passing (Fig.3.2.78): passing with cut (See Section3.2.12.7.1 on p. 949), passing via fillet (See Section3.2.12.7.2 on p. 950), passing via fillet with equal radius (See Section3.2.12.7.3 on p. 950).

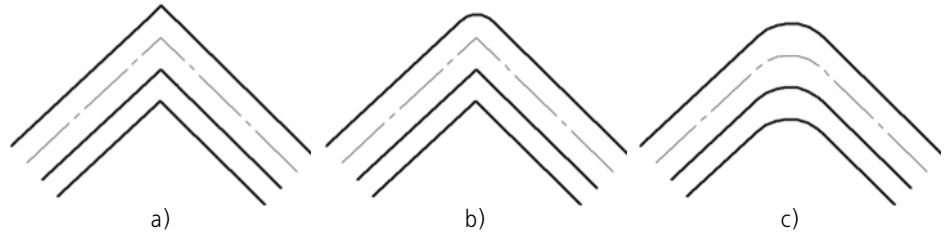


Fig. 3.2.78. Methods of passing the corner in the base line vertex: a) passing with cut, b) passing via fillet c) passing via fillet with equal radius

- ▼ When creating a multiline, the method of vertex tracing is defined for the start vertex of the segment being added. The defined method is used by default for the subsequent vertices.
- ▼ When editing a multiline, you can define the required method of tracing for any vertex of that multiline except the end vertices.



To select the corner passing method use the **Corner Passing** group of switches on the **Multiline** tab of the Properties toolbar.



To define the required corner passing method enable the required switch: **With cut**, **With fillet**, **Via fillet with constant radius**.




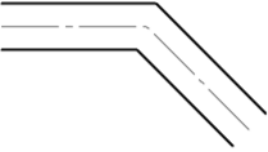

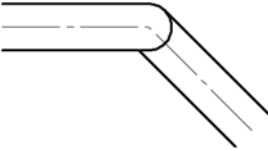
3.2.7.1. Passing with Cut




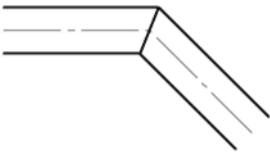

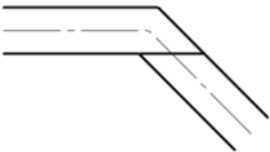
The **Passing with Cut** method allows to insert the separator in the joint of the multiline segments.

To select the required separator, open the **Separator** list in the **Multiline** tab of the Properties toolbar and specify the required line in it (Table 3.2.4).

Tab. 3.2.4. Separators of multiline segments

Delimiter	Symbol
	Without separator 
	Arc 

Tab. 3.2.4. Separators of multiline segments

Delimiter	Symbol
	<p>Angular</p> 
	<p>Overlapping</p> 

For the **Arc** and **Overlapping** separators you can define a direction, i.e. select the one of the two segments to be joined for which a separator will be created.

To select the separator direction use the **Direction** group of switches.

To define the required separator direction enable the required switch: **Direction 1** or **Direction 2**.

The drawing up line of the separator has the *Normal* style by default. Use the **Style** list on the Properties toolbar to change the line style. The list is unavailable if there is no limiter at the segments joint.

Please, note that if the drawing up style of the arc separator differs from the style of outermost lines of the multiline, the joint of the segments will look as shown on Fig. 3.2.79.

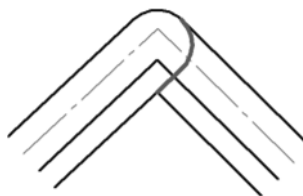


Fig. 3.2.79. Segment joint with the arc separator

3.2.12.7.2. Passing via fillet



The **Passing via Fillet** method connects the multiline lines in the segment joints by rounding them off with the equidistant radius.

3.2.12.7.3. Passing via Fillet with Equal Radius



The **Passing via Fillet with Equal Radius** method connects the multiline lines in the segment joints by rounding off. The fillet radius is the same for all lines. By default, the fillet radius is equal to 10.

Enter the required fillet radius in the **Radius** field or define it using the counter.

3.2.12.8. Multiline limiters

The default view and settings of limiters are defined during the multiline setup (see Section 9.2.6.9 on p. 1770). You may change the view of the multiline end segment limiter (See Section 3.2.12.8.1 on p. 951) and its settings: the height and style of the drawing up line (See Section 3.2.12.8.2 on p. 951).



After editing the limiters may stop displaying in the document. This occurs in the following cases:


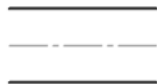

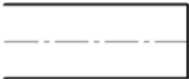

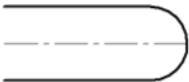

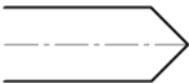
- ▼ one line has been left in the multiline;
- ▼ neither outermost line is displayed in the multiline.

Information of the non-displaying limiters is saved in the multiline.

3.2.12.8.1. Selection of the multiline limiter view

To define a limiter view, define the required line on the **Limiter 1** (or **Limiter 2**) line in the **Settings** tab of the Properties toolbar (Table). 3.2.5).

Tab. 3.2.5. Multiline limiters

Limiters	Symbol
 Without limiter	
 Straight-line	
 Arc	
 Polyline	

3.2.12.8.2. Definition of the limiter settings

To define the limiter height, enter its value in the **Height 1, %** (or **Height 2, %**) field of the **Settings** tab of the Properties toolbar. This parameter's value is set as percentage, from 0 to 100, of the multiline's width.

To change the line style, use the **Style 1** (or **Style 2**) list on the Properties toolbar.

Please note that if the drawing up style of the straight-line limiter differs from the style of the multiline outermost lines, depending on the height value the limiter will look as shown in Fig. 3.2.80.



Fig. 3.2.80. Straight-line limiter:
a) with zero height; б) with the height greater than zero

3.2.12.9. Multiline Operation Modes



During the multiline creation the segment adding mode is enabled by default — the **Add Segments** button on the Special Control Panel is pressed. In this mode a base line is created, multiline settings are defined and methods of passing the corner in the multiline vertex are selected (See Section 3.2.12.3 on p. 942).

The second mode of working with the multiline— item editing mode. In this mode you may change position of the base line vertices (see Section 3.2.12.10.1 on p. 953) and insert new vertices in it (see Section 3.2.12.10.2 on p. 954). In this mode you may also change multiline settings (See Sections 3.2.12.4 on p. 943, 3.2.12.7 on p. 948 and 3.2.12.8 on p. 951). To switch to the editing mode, press the **Edit items** button on the Special Control Panel. Vertices of the base line in the form of small black squares appear on the multiline ghost.



The **Edit Items** button is available if the multiline has at least one segment.

You may switch between the modes of segments adding and editing using the Special Control Panel buttons or commands in the context menu.

By default, when you edit the already created multiline, the element edit mode is enabled automatically.

3.2.12.10. Vertex editing

When needed, you may reconfigure the base line of the multiline by editing its vertex.



Base line vertices shall be edited in the multiline element edit mode (the **Edit Elements** button is pressed on the Special Control Panel). The multiline operation modes are detailed in Section 3.2.12.9 on p. 952.

The group of switches **Operation** (Fig. 3.2.81) allows to select the baseline vertex edit operation:



- ▼ **Vertex Editing** is a change in position, parameters or deletion of a vertex (see Section 3.2.12.10.1 on p. 953);



- ▼ **Vertex Insertion** (see Section 3.2.12.10.2 on p. 954).

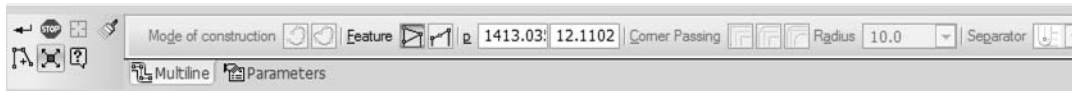


Fig. 3.2.81. The group of switches **Operation**

3.2.12.10.1. Vertex editing operation

Vertex editing operation allows:

- ▼ changing the vertex position;
- ▼ removing the vertex;
- ▼ defining the method of passing the corner in a vertex, if this vertex is not an end one (See Section 3.2.12.7 on p. 948).



To edit a vertex, enable the **Edit Vertices** switch in the **Operation** group.

Vertex defining



Move the cursor to the required vertex. The cursor shape will change—it will turn into a four-pointed arrow. Left-click the mouse. The vertex will be activated—its color will change from black to that defined for highlighted objects (see Section 9.1.7.11 on p. 1713).



To cancel the selection, click outside the multiline ghost.

Vertex moving

Define a vertex. Change its position moving the mouse or defining new coordinates (See below). Besides, for this vertex you may define the required method of corner passing (See Section 3.2.12.7 on p. 948).

Defining vertex coordinates

Define a vertex. On the Properties toolbar beside the **Operation** group there are fields with coordinates of this vertex. Enter new values in the coordinate fields and fix them by pressing the **<Enter>** key. After that the highlighted vertex will take a new position.

Vertex removal

Define the vertex and press the **<Delete>** key. The vertex will be removed and the multiline will be reconstructed.

- ▼ During the removal of the multiline end vertex, the segment corresponding to it will be removed.
- ▼ During the removal of any other vertex of the multiline the segments adjoining it will be removed. A straight-line segment will be inserted in the multiline instead.

After the removal of one-segment multiline vertex the system will switch to segment adding mode.

3.2.12.10.2. Vertex Insertion operation



Operation of the vertex insertion into the base line adds a new vertex to the base line.

To add a new vertex to the base line, enable the **Insert Vertex** switch in the **Operation** group.

Insert a new vertex into the base line.

You may define the insertion point for the new vertex with the cursor on the multiline ghost. If the indicated point does not belong to the multiline, the position of the new vertex will be defined by the projection of a point on the base line.

To create a new vertex enter its coordinates into the fields located in the Properties toolbar. The entered coordinates should be confirmed by pressing the *<Enter>* key.



For the new vertex the current corner passing method will be used (see Section 3.2.12.7 on p. 948).

3.2.12.11. Deleting and Restoring segments of multilines

After multiline creation, the **Trim Line** and **Trim Line by 2 Points** commands can be used for its lines (see Sections 3.4.7.1 on p. 1137 and 3.4.7.2 on p. 1138). As a result of executing these commands, the multiline segments will be deleted.

If necessary, you can restore deleted segments of multilines by two methods.



1. To restore one or several deleted segments, call the **Restore deleted multiline's segment** button from the **Editor** menu or press the same-name button on the **Editing** toolbar.

Move the cursor to one of the outermost points of the segment that should be restored. Selected portion will be highlighted. After the mouse click, this portion will be restored. You can restore arbitrary number of portions by sequentially selecting them with cursor.

If you need to restore several deleted segments of multilines or several multilines, it is convenient to use selecting by marquee frame. To select a portion using this method, it is sufficient that one of its end points was inside marquee frame. All segments specified by the marquee frame are highlighted and restored after the frame is formed.



2. To restore all deleted segments of multiline, call the **Restore Multiline Line Integrity** button from the **Editor** menu or press the same-name button on the **Editing** toolbar.

Point the cursor to the multiline line which integrity you need to restore. Selected line will be highlighted. After the mouse click, all the deleted portions of this line will be restored. You can restore integrity of arbitrary number of portions by sequentially selecting them with cursor.

If you need to restore integrity of all lines of a multiline or several multilines, it is convenient to use specification by a cutting segment (see Section *Specifying a group of objects with a cutting segment* on page 357). The selected lines are highlighted. After forming cutting segment, deleted portions of all selected lines are restored.

Multilines that contain deleted portions can be selected in the document window before calling the Restore Multiline Line Integrity command. In this case, a prompt to restore all deleted parts of these multilines appears on the screen after calling the command. To restore the deleted segments, click the **OK** button. The **Cancel** button serves to reject the restoring.

Further working with the command is described above.

3.2.12.12. Multiline editing features

Use all editing commands except the **Align along Border**, **Extend to Nearest Object**, **Clear Area**, **Transform into NURBS** commands. See a detailed description of editing commands in Section 3.4 on p. 1113.



The multiline may be used as a limiting object in commands **Align along Border** and **Extend to Nearest Object**.



Besides, you may split the multiline into separate objects. For this use the command **Editor – Destroy**.



The command **Remove Facet or Fillet** (See Section 3.4.7.5 on p. 1140) removes an arc or straight-line segment of the multiline provided that the remained segments may be extended up to the intersection point.

3.2.13. Hatch and Fill

Hatching and filling areas of a graphic document with color in KOMPAS-3D is similar to the way you build objects.

To create a hatch or fill, you have to specify its border and parameters. The methods of specifying borders, which are the same for hatches and fills, are described in Section 3.2.13.1, techniques of creating hatches— in Section 3.2.13.2 on p. 958, techniques of creating fills — in Section 3.2.13.3 on p. 959.

3.2.13.1. Specifying Borders

The number and numbering level of areas specified at the same time are unlimited.

The objects limiting the hatch (fill) can be selected before the command is executed. In this case, after you execute the command, the system will display a request for the use of selected objects as the hatch (fill) boundary. To confirm, click **Yes**, and the system will immediately build a hatch (fill), if possible.

If no objects were selected or if you chose not to use them, specify a point within the area you want hatched (filled). The system will determine the immediate border within which the point has been specified and create a hatch (or fill) ghost.



The borders of hatch and fill may be system lines of styles *Normal* and *Broken Line* or user style lines during configuring which it was defined that they can be used as hatch borders.

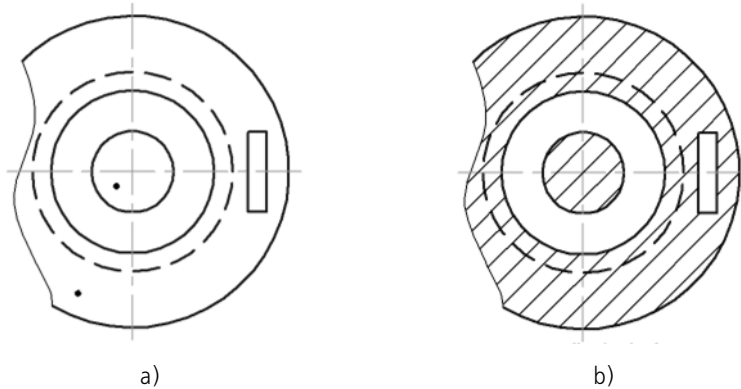


Fig. 3.2.82. Constructing a hatch:
a) points within the areas selected for construction, b) resulting hatch

Specifying a point within an area is the default way to specify the hatch (fill) border. You can also form the border manually or with an arrow. These methods are detailed below.



When you create associative hatches or fills (for associative objects see Section 7.2.1.4 on p. 1602) the modes of drawing borders manually or tracing borders by arrow are inaccessible.

In addition, you can specify the hatch (fill) of existing geometric objects as a border for a new hatch or fill. The commands **Add Border** and **Remove Border** in the context menu are intended for this purpose. Each of them opens a submenu with the list of ways to specify objects.



If the hatch (fill) in several non-connected areas of the document should have similar settings, it should be created as a single object.

For example, the borders of **all hatch areas** of a part section (Fig. 3.2.83) should be specified as borders of the **same hatch**. This will ensure the hatch parameters inside them are identical and simplify further editing.

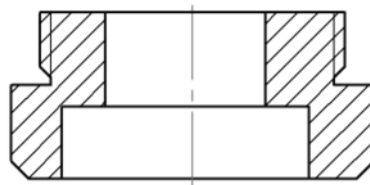


Fig. 3.2.83. Hatch consisting of several areas

3.2.13.1.1. Manual Drawing of Borders

Drawing borders manually means specifying manually the border of an area where a certain operation will be performed. Borders drawn in this way are displayed only until the command is completed.



To switch to the manual border drawing mode, press the button **Manual Drawing of Borders**.

The set of control elements on the Property bar will change; it will look as shown in Fig.3.2.84.



Fig. 3.2.84. Property bar in the manual border drawing mode



Sequentially select polyline vertices bounding the region.

Switches of the **Mode** group let you control the closing of a polyline.

You can change the position of polyline vertices directly in the process of drawing. To do this, click the **Edit Points** button.



Move the pointer to any handle (displayed as black squares). Shape of the cursor will change: it will turn into a four-sided arrow.



Change the position of handles by an way or delete unnecessary points, similar to editing a polyline (see Section3.2.9.1.2 on p. 924)



You can also add handles. To do this, click the required segment. The selected segment will be divided into two pieces by the new vertex that will be positioned at the specified point. You can “drag” it to any position.

To exit the mode of editing the handle positions, release the **Edit Points** button.

When done drawing the border, press the **Create Object** button.



To cancel manual drawing of the border, press the <Esc> key or the **Interrupt Command** button.

3.2.13.1.2. Trace Border by Arrow

Tracing the border by arrow means specifying the border of an area where you want to perform any operation by tracing intersecting geometric objects.



To switch to the mode of border tracing by arrow, click the button **Border by Arrow Direction**.

The set of control elements on the Property bar will change; it will look as shown in Fig.3.2.85.



Fig. 3.2.85. Property bar in the mode of border tracing by arrow

Select a point near a geometrical object, from which you need to start traversing the contour.

A ghost image of the first contour segment is displayed (overlaid on the base element), and an arrow indicating the system-proposed direction to move in.

To define the necessary direction, click with the mouse on the appropriate object.

You can also select the direction of movement by contour segments using the following buttons on the Special Control panel:



▼ **Previous Direction,**



▼ **Next Direction,**



▼ **Step Forward,**





▼ **Step Backward.**

In addition, you can change the movement direction using keyboard shortcuts listed in Table 3.2.6.


Tab. 3.2.6. Keyboard shortcuts for selecting the tracing direction


Key	Description
<Space>	Choosing among the possible continuation directions from the current node counterclockwise.
<Shift>+<Space>	Choosing among the possible continuation directions from the current node clockwise.
<Enter>	Confirmation of the choice of direction marked with the arrow.
<Shift>+<Enter>	Go one step back.


 Group of switches **Node Passing Mode** lets you specify in what way you will pass nonbranching nodes of the contour, i.e., nodes in which there is only one movement direction (meaning that the node has no branches). By default, automatic processing of such nodes is used (without a prompt to select the direction to continue).


 To reject tracing the border and start forming the contour anew, click **Reselect** and specify the required object from which you want to start tracing.

When you specify a section that closes the contour, i.e., a section whose end point coincides with the start point of the first section, the object becomes fixed automatically.


 To fix an open contour, click the **Create Object** button.


 An open contour existing in a graphic document can be selected as a hatch border only using the method of border tracing by arrow.

3.2.13.2. Hatch

3.2.13.2.1. Creating Hatches



You hatch one or several areas, click **Hatch**.
Specify hatch borders and parameters.





The hatch ghost will appear, enabling you to control whether you have selected hatch areas and parameters correctly.

When all the required areas are hatched, click the **Create Object** button. The hatch will be fixed in the document, and the system will wait for you to specify the border for the next hatch.

3.2.13.2.2. Hatch Parameters

Hatch parameters can be configured using control elements on the **Hatch** tab of the Property bar. They are represented in Table 3.2.7.

Tab. 3.2.7. Hatch control elements

Element	Description
Style	List of hatch styles. To change the current hatch style, open the list and select the required style.
Color	The list lets you select the color of hatch (or fill) lines. Clicking the Another Color item displays the advanced dialog of the color selection.
Base point	A conventional start point for the hatch. When changing its coordinates, hatch lines move vertically or/and horizontally.
Step	Hatch spacing.
Angle	Hatch rotation angle around the datum point.
Type	
	Hatching the entire area inside the border.
	Creating a hatch as a band along the border.
Width*	Value of the hatch band thickness.
Position*	
	Location of the hatch band to the left of the border.
	Location of the hatch band to the right of the border.

* This control element is available if the **Band** switch is active in the **Type** group.

3.2.13.3. Fill

3.2.13.3.1. Fill Types

Fills of the following types can be created in KOMPAS-3D.

- ▼ **Monochromatic** fill representing a solid color fill of a selected area.
- ▼ **Gradient** fill with a gradual or step-by-step color variation, comes in several types (linear, cylindrical, angular, conical, radial and square) and gives a third dimension effect to images in a graphic document (see Fig. 3.2.86).
- ▼ The fill area can consist of one or several closed contours, including disconnected contours.

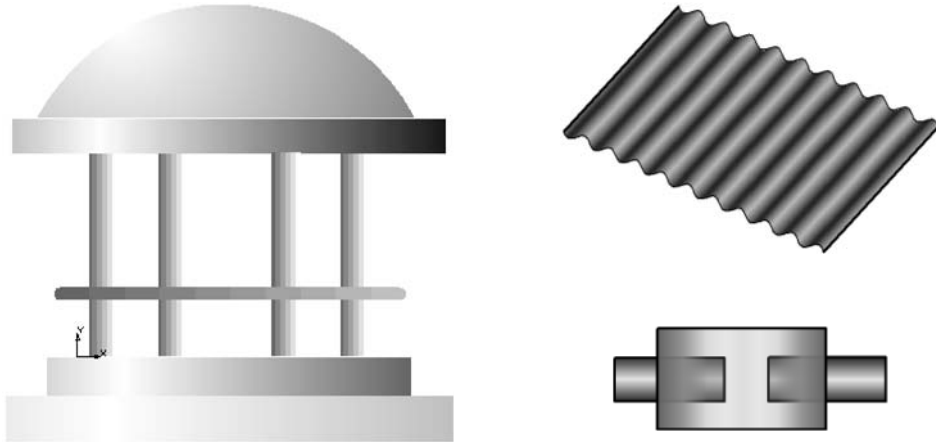


Fig. 3.2.86. Example of using fills

The following specifics have to be considered when creating gradient fills.

After you have specified one or several fill areas around contours, the system automatically creates a **dimensional rectangle** inside which color distribution will occur. When you change fill parameters, such as the fill angle, the dimensional rectangle will also change (see Fig. 3.2.87). The dimensional rectangle is conventional and is not displayed on the screen.

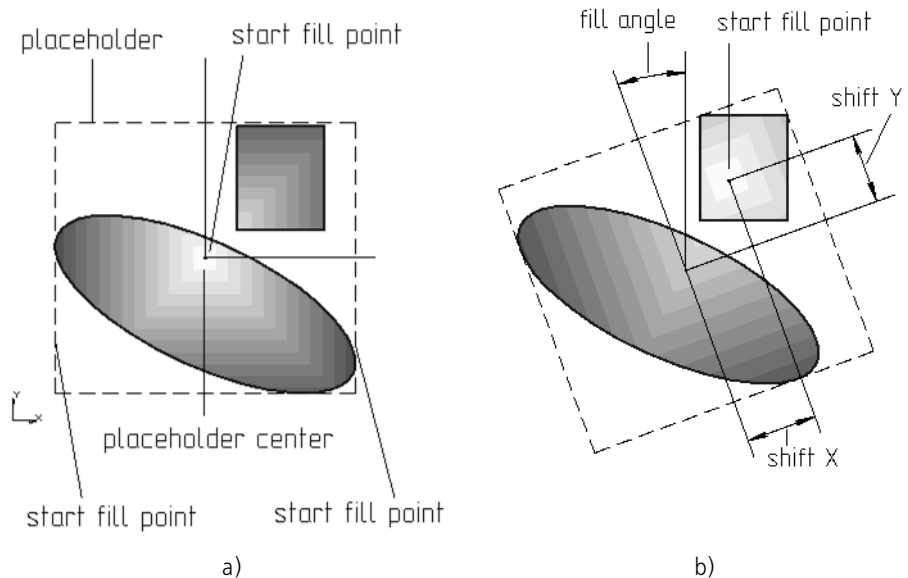


Fig. 3.2.87. Fill parameters when drawing:
 a) in the absence of shift or angle, b) after specifying shift and angle

The **start point** of a fill is a point where you specify the start color. By default, the start point is at the center of the dimensional rectangle.

The **start fill line** is a line passing through the start point. Color graduation for certain fill types begins from the start line or ray.

The **end line** of the fill is an end line or ray in which the end color is set.

The Table 3.2.8 describes gradient fill types. Examples with step-by-step color variation, zero shift of the start point and zero fill angle are provided for illustrative purposes.

Tab. 3.2.8. Description of gradient fill types


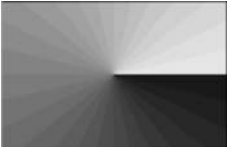


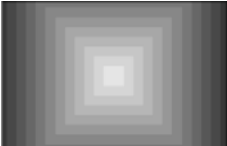
Sample fill
 (with step-by-step
 color variation)



Fill type description

Linear— color changes from the start line to the end line. The start and end lines are on parallel sides of the dimensional rectangle. Linear fill has not start point.

Tab. 3.2.8. Description of gradient fill types

Sample fill (with step-by-step color variation)	Fill type description
	Cylindrical — color changes from the start line to end lines. The end lines are sides of the dimensional rectangle parallel to the start line.
	For angular and conical fills, the start and end lines are rays drawn from the start point. Angular — color changes from the start ray to the end ray in a circular direction. The distance between the start ray and the end ray is 360 degrees.
	Conical — color changes from the start ray to the end ray in a circular direction. The distance between the start ray and the end ray is 180 degrees.
	Radial — color changes in a radial direction from the start point to a most distant point on the side of the dimensional rectangle.
	Square — color changes in directions perpendicular to the sides of the dimensional rectangle. The start color is in the start point, the end color is on the most distant side of the dimensional rectangle.

Fill angle determines the angle of fill rotation relative to the start point inside the fill area. When you change the fill angle, the dimensional rectangle will also rotate by the specified angle. For linear fill, the angle determines the change in the direction of color variation.

The position of the start point of the fill can be changed by the length of the **shift** relative to the center of the dimensional rectangle. If there is no shift, the start point is at the center of the dimensional rectangle.

If necessary, you can increase the **transparency** of each fill color. 100% transparency means that the object is invisible. By default, the values of “Transparency” (“Start Transparency”, “End Transparency”) are 0%, meaning that fills are created non-transparent.

Several intermediate colors and transparency levels may be additionally defined in the fill.

3.2.13.3.2. Creating a Fill



To fill one or several areas, click **Fill**.

Specify fill borders and parameters.



To copy the parameters of a fill previously created in a document, click the **Copy Properties** button on the Special Control panel.

The cursor will change in appearance.



Move the cursor to the contour limiting the fill whose parameters you want to copy. Once the face is highlighted, press the left mouse button. To move the parameters, specify the fill area. The fill with copied parameters will appear in the selected area. If necessary the text can be edited.

One or several filled areas will appear on the screen, letting you control how you specify fill areas and parameters.



When all the required areas are hatched, click the **Create Object** button. The fill will be fixed in the document, and the system will wait for you to specify the border for the next fill.

3.2.13.3.3. Fill Parameters

Fill parameters can be configured using control elements on the **Fill** tab of the Property bar.




The set of control elements depend on the fill type selected in the **Type** group.

They are represented in Table 3.2.9.

Tab. 3.2.9. Fill control elements

Element	Description
Type	List of fill types. To change the current fill type, open the list and select the required type.
If you select the monochromatic fill type, the following control elements are available.	
Color	The list lets you select the fill color. Clicking the Another Color item displays the advanced dialog of the color selection.
Transparency	Control element that lets you set transparency* of the selected color. <ul style="list-style-type: none"> ▼ If Zero-level Transparency% 0% is set, the fill is completely non-transparent. To increase transparency, move the slider to the right. ▼ The slider in the rightmost position sets Transparency to 100%. The fill will be completely transparent.
If you select other fill types in the Type list (gradient: linear , cylindrical , etc.), the following control elements are available.	
Color 1	The lists that enable the selection of the start and end colors of the fill.
Color 2	Clicking the Another Color item displays the advanced dialog of the color selection.

Tab. 3.2.9. Fill control elements

Element	Description
Angle	<p>Control element that lets you specify the fill rotation angle relative to the start point inside the area.</p> <p>Enter a value or select one from the list of previously entered values and frequently used angles.</p> <p>This field is not available for the radial fill.</p>
X-axis Offset Y-axis Offset	<p>The fields for entering or presetting the relative shift of the fill start point from the center to the sides of the dimensional rectangle with a counter. Values can vary between -100 and 100%.</p> <ul style="list-style-type: none"> ▼ If the value is set to 0%, there is no shift — the fill start point coincides with the center of the dimensional rectangle. ▼ The value of 100% for Shift X or Shift Y means that the fill start point is on the side (contour) of the dimensional rectangle. <p>When you move a point with the mouse or rotate the fill, the shift value is calculated and entered into these fields.</p> <p>For cylindrical fill, the Shift field is available for entering the shift value of the start point in the direction toward the start or end line of the fill. These fields are not available for the linear fill.</p>
Transparency	<p>The control elements which are used to specify the transparency* of the selected Color 1 and Color 2.</p> <ul style="list-style-type: none"> ▼ If you set Start Transparency (End Transparency) to 0%, the fill specified in fields Color 1 (Color 2) is completely non-transparent. To increase the transparency of a specific fill color, move the relevant slide to the right. ▼ The slider in the rightmost position sets Start transparency (End transparency) to 100%. The fill will be completely transparent.
 Advanced	<p>When pressing the key, the advanced fill control dialog will be displayed. The dialog defines intermediate colors and transparency levels as well as their location in the fill (See Section 3.2.13.3.4).</p>
Color Gradients	<p>The group of switches enabling the selection of a method of the fill color changing from Color 1 to Color 2.</p> <ul style="list-style-type: none"> ▼ If the switch Smooth Color Gradient is active, the color gradient will be smooth. ▼ If the switch Specified Number of Steps is active, the color gradient will match a specified number of steps. The field for entering or specifying the number of steps with a counter becomes active, from 1 to 255.
	
	

* Some printers do not support printing of transparent fills. In these cases when customizing printing it is recommended to enable an alternative way of printing (see Sections 8.1.2.2 on p. 1634 and 8.2.5.2 on p. 1652).

3.2.13.3.4. Additional color gradients and transparency levels of a fill

When building or editing gradient fill you may create **additional** intermediate colors and/or change transparency level of some of its area (See Fig.3.2.88).



For this, press **Advanced** button on the Properties button and call the context menu command with the same name.

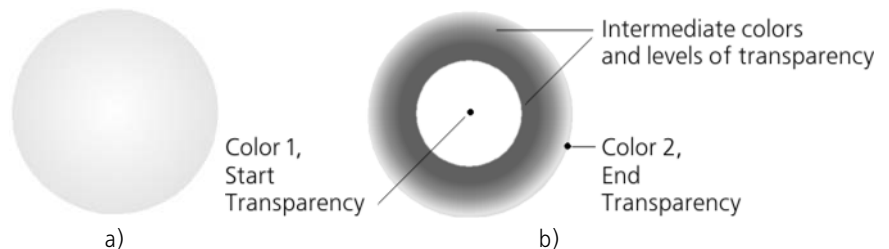


Fig. 3.2.88. Radial fill before and after the adding several colors and levels of transparency

The dialog of advanced fill control will be displayed (See Fig.3.2.89).

In the dialog window there is a rectangle sample with the color gradient defined on the Properties toolbar from **Color 1** to **Color 2** and transparency levels — **Start** and **End**. If non-zero transparency was defined on the Properties toolbar, in the upper half of the rectangle black-and-white cells will be displayed the brightness of which will help assess the transparency level.

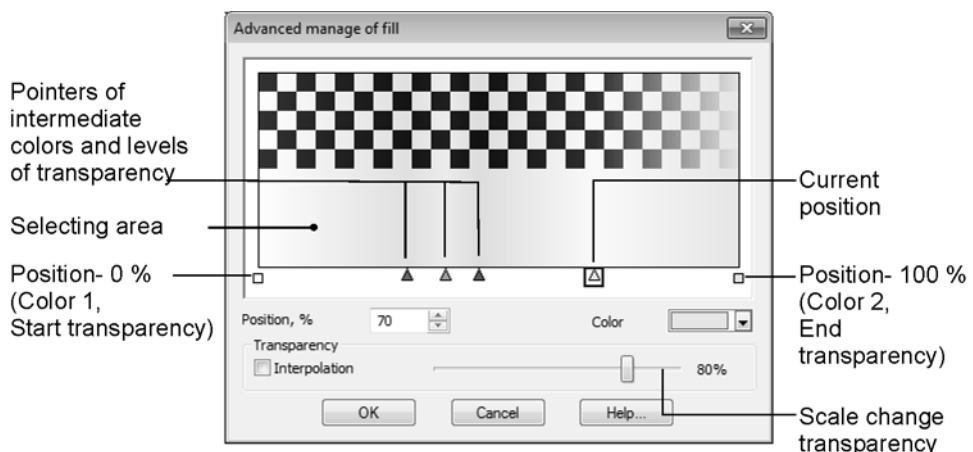


Fig. 3.2.89. Advanced fill control dialog

To define additional colors and/or transparency levels, do the following.

1. Create a new position. In this dialog window double click on the location of a new color or transparency level between **Color1** and **Color2**. In the bottom of the window a triangle-type item pointer will appear.

You can create several positions.

The current position pointer will be displayed in the frame (See Fig. 3.2.89). Change the current position with the mouse click on the required pointed or using the key combinations `<Ctrl> + <▶>`, `<Ctrl> + <◀>`.

2. Move the pointer to the required position with the mouse or keys `<▶>`, `<◀>` or by the input or counter adjustment of the integer from 1 to 99 in the **Position** field. When moving with the mouse the cursor will switch to two-sided arrow.



In the **Position** field a value in percentage will be displayed showing the current position location relative to the original fill point. Initial color corresponds to position 0, end fill color corresponds to 100% .

The **Position** field is available if the pointer of additional position is the current one.

3. Define the color in the current position in the color selection dialog clicking twice the position pointer or using the **Color** control. By default the color in the created position is an intermediate one between the colors in the nearest positions.

▼ To create an area in the fill with the constant color, move the position pointer with the mouse to the required distance with the `<Ctrl>` key pressed.

▼ To remove the intermediate color, “drag” this pointer with the mouse outside the window. After the cursor changes color, release the pointer. You can also delete the intermediate color pressing the `<Delete>` key after the relevant position has been accepted as the current one.



If you need to enter additional colors without changing the transparency, the **Interpolation** option of the **Transparency** group must be enabled.

4. Define additional transparency levels in the fill. To do this, make the pointer the current one at the border of the area where transparency will change.

The **Interpolation** option in the **Transparency** group controls transparency presets, and the scale with the slider is used — to define and display transparency level in the current position. The transparency value is shown in percentage to the right from the scale.

▼ By default the **Interpolation** option is enabled. This means that the transparency in the created position is a value interpolated between the transparency values of the neighboring positions, for example, the start and the end one.

It is impossible to randomly change the transparency in the current position— changing scale is inaccessible.

▼ If the **Interpolation** option is disabled, the transparency changing scale will become accessible. To define a new value, move the slider to the required position with the mouse or keys `<▶>` and `<◀>`.

If the transparency value is equal to 0%, the fill in this position is considered to be fully non-transparent, if it is 100%— it is fully transparent. The color in this case fully corresponds to the color displayed in the **Color** control field.

To cancel the additional transparency level without the position deletion, make the position as the current one and enable the Interpolation option.

After defining intermediate colors and transparency levels press **OK**.



The start and the end color, as well as the start and the end transparency of the fill may be changed making positions 0% and 100% the current ones. After the changes has been effected, the values will automatically be displayed on the Properties toolbar.

3.2.13.3.5. Editing the Fill

Fill handles are the start point and fill contour handles.

Bear in mind the following when editing the fill.

- ▼ In the mode of editing handles for all fill types, except monochromatic and linear fill, you can change the position of the start point.
- ▼ To prevent the fill color from changing when highlighted, enable the option **Always Show Fill in Specified Color** in the dialog that appears after you click **Service— Options— System— Graphic Editor— Edit**. In this case, only the fill contour will be highlighted when you select it.

3.2.14. Complex Objects

In KOMPAS-3D, complex geometric objects include:

- ▼ offset curves,
- ▼ contours.

A common property of these objects is the fact that they are selected and edited in their entirety. You can work with individual objects making up such a complex feature only after exploding it. To explode an object, click **Edit — Explode**.

3.2.14.1. Contour



To create a contour consisting of parts of intersecting geometric objects, click **Create Contour**.

Further steps are identical to those described in Section 3.2.13.1.2 on p. 957.

The control of the source objects is described in Section 3.4.2.1.1 on p. 1121.

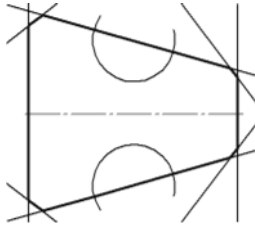


Fig. 3.2.90. Sample of contour created through tracing by arrow

3.2.14.2. Equidistant Line



To draw an equidistant line of any geometric object, click **Equidistant Line**.

Specify the object whose equidistant line you want to create. The equidistant line ghost will be displayed.

Configure the equidistant line settings (see Section 3.2.14.2.1). All changes you make are reflected in the ghost.







Once you complete settings, click the **Create Object button**. The equidistant line will be fixed in the document, and the system will wait to specify the border for the next equidistant line.





3.2.14.2.1. Equidistant Line Parameters

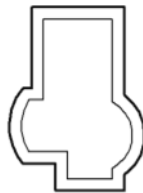
Equidistant line parameters are configured using items on the **Equidistant Line** tab of the Property bar. They are represented in Table 3.2.10.

Tab. 3.2.10. Equidistant line control elements

Element	Description
Radius 1	Distance from the object to the equidistant line on the left.
Radius 2	Distance from the object to the equidistant line on the right.
Corner Passing	
	Method of passing corners with cut (Fig.3.2.91, a).
	Method of passing corners via fillet (Fig.3.2.91, b).
Type	
	Creating an equidistant line on the left of the object.
	Creating an equidistant line on the right of the object.

Tab. 3.2.10. Equidistant line control elements

Element	Description
	Building an equidistant line on two sides of an object (Fig.3.2.92, b).
Mode	
	Remove degenerated portions of an equidistant line (Fig.3.2.93, a).
	Retain degenerated portions of an equidistant line (Fig.3.2.93, b).
	If the Retain Degenerated Portions switch is active, the corners are passed only with cut. The group of switches Corner Passing is unavailable.



a)



b)

Fig. 3.2.91. Corner passing: a) with cut, b) via fillet



a)



b)

Fig. 3.2.92. Equidistant line: a) on one side of the object, b) on both sides



Fig. 3.2.93. Equidistant line: a) without degenerated portions, b) with degenerated portions

3.2.14.3. Equidistant Line by Arrow



To draw an equidistant line of a contour formed through tracing by arrow, click **Equidistant Line by Arrow**.

The steps for forming a contour through tracing by arrow are described in Section 3.2.13.1.2 on p. 957.

The parameters of an equidistant line by arrow are the same as in the case of an equidistant line of an object (see Section 3.2.14.2.1 on p. 968).

3.2.15. Chamfers and Fillets

By default, an arbitrary chamfer and fillet are created using the line style of the first of the two objects. A chamfer or fillet on object corners have the same style as the object.

3.2.15.1. Chamfer



To draw a line connecting two intersecting curves, click **Chamfer**.



By using group of switches **Type** on the Property bar, specify the chamfer construction method: by two sides or by a side and angle.



Enter chamfer parameters in the fields on the Property bar.

Select the first and the second object, between which you need to draw a chamfer.

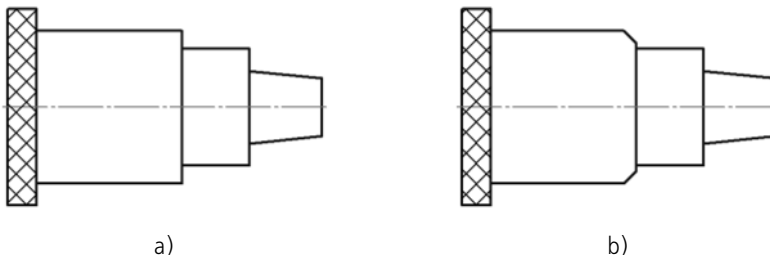
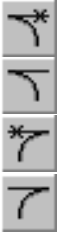


Fig. 3.2.94. Creating a chamfer: a) original image, b) result of the command

3.2.15.1.1. Controlling the Object Trimming

There are two methods of creating a chamfer/fillet:

- ▼ with object trimming,
- ▼ without object trimming.



When using the first method, the pieces of the first/second object remaining after creating a chamfer/fillet, are automatically deleted; when using the second method, they remain.

You can choose the method for creating a chamber for both its elements individually. To do this, in the groups **Element1** and **Element2** on the Property bar, activate the switch **Trim Element** or **Do Not Trim Element**. By default, both of them has the **Trim Object** switch active.

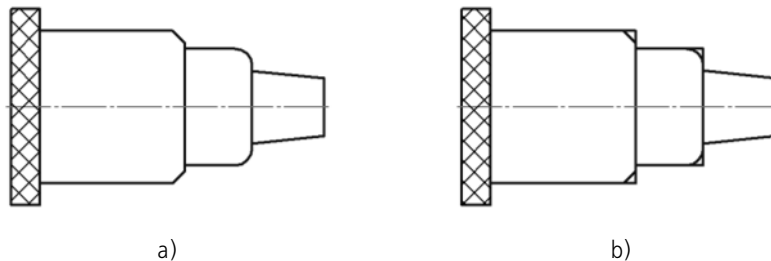
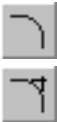


Fig. 3.2.95. Creating a Chamfer/Fillet: a) with element trimming, b) without element trimming

3.2.15.1.2. Creation of the Intersection Notation of Truncated Objects

An intersection notation can be created on trimmed objects. It means that when creating a chamfer/fillet, objects will be extended by thin lines to their intersection point. An intersection notation can be also constructed separately by using the **Intersection Notation** command, see Section 3.3.12.19 on p. 1065.



In order to enable or disable creation of the intersection when creating a chamfer/fillet, enable the required switch in the **Intersection Notation** on the Property bar.

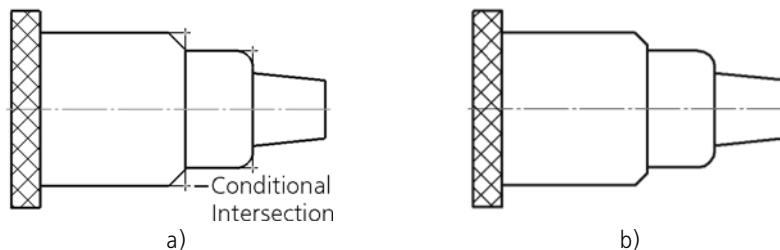


Fig. 3.2.96. Creating a Chamfer/Fillet: a) if the intersection creation is enabled, b) if disabled

The intersection will be created, if the chamfer/fillet is created with trimming of one of the specified objects (see Section 3.2.15.1.1 on p. 971).

Default parameters values defined in the **Intersection Notation** section of the settings dialog are used for constructing (see Section 9.2.6.12 on p. 1785).

By default the created cosmetic intersection is in no way connected to the objects, specified during the course of the command, and can be edited separately.

3.2.15.2. Chamfer on Object Corners



To create chamfers on corners of the following object types: a contour, polyline or polygon, click **Chamfer on Object Corners**.



By using group of switches **Type** on the Property bar, specify the chamfer construction method: by two sides or by side and angle.



Enter chamfer parameters in the fields on the Property bar.

Select a corner of a contour, a polygon or a polyline to create a chamfer. The first object used in building a chamber will be the segment closest to the specified point.



By default, a chamfer is built only on the specified corner of the contour. In this case, the **At Selected Corner** switch is active in the **Mode** group on the Property bar. In order to create chamfers with specified parameters simultaneously at all corners of a selected contour, activate the switch **At All Contour Corners**.



activate the switch **At All Contour Corners**.



Fig. 3.2.97. Building chamfers on all corners of a rectangle

3.2.15.3. Fillet



To draw a fillet by circle arc between two intersecting objects, click **Fillet**.

Enter the fillet radius in the respective field on the Property bar.

Select to objects between which you need to construct a fillet.

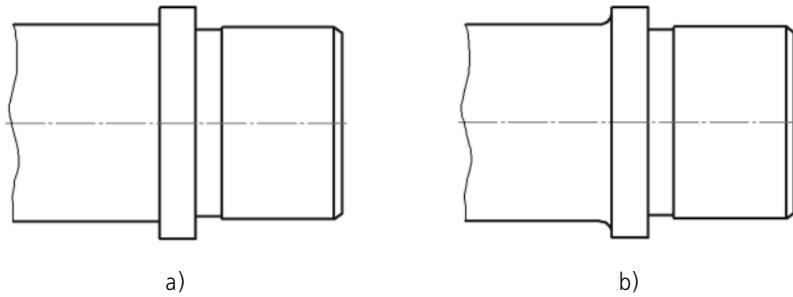


Fig. 3.2.98. Building a fillet: a) initial image, b) result of building a fillet

3.2.15.4. Fillet Object Corners



To create a fillet by circular arcs on the corners of the following type of objects: a contour, polyline or polygon, click **Fillet Object Corners**.

Enter the fillet radius in the respective field on the Property bar.

Select a corner of a contour, a polygon or a polyline to create a chamfer.



By default, only the specified contour corner will be rounded. In this case, the **At Selected Corner** switch is active in the **Mode** group on the Property bar. For fillets with specified parameters to be created simultaneously at all corners of a selected contour, activate the switch



At All Contour Corners.

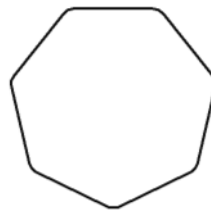


Fig. 3.2.99. Creating fillets on all vertices of a polygon

3.3. Dimensions and designations

3.3.1. General Dimension Handling Techniques

KOMPAS-3D allows creation of any of the standard dimension variants in a graphic document. It is possible to indicate several types of linear, angle, radial, diameter dimensions, as well as height and arc dimensions. Also, a special method of dimensioning is available, in which the system indicates dimensions automatically.

The dimensioning commands are grouped in the menu **Tools – Dimensions**, and the command calling buttons are located on the **Dimensions** bar (Fig. 3.3.1).

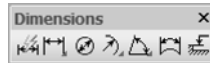


Fig. 3.3.1. **Dimensions** bar

The general sequence of steps while dimensioning is:

1. Calling of the command for a specific type of dimension or the automatic dimensioning command.
2. Specifying the object(s) to dimension.
3. Setting up the dimension line view using the Settings bar tabs (see Sections 3.3.1.1 and 3.3.1.2).
4. Editing (if needed) the dimension text (see Section 3.3.1.3) and specifying its position.

3.3.1.1. Property Settings



Fig. 3.3.2. Usage of the **Dimension** tab of the Property bar while specifying a linear dimension

The dimension control widgets located on the **Dimension** tab are presented in Table 3.3.1.






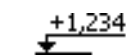
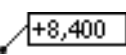
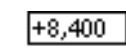



Tab. 3.3.1. Control elements on the **Dimension** tab

Element	Description
p1, p2, p3, p4	Handle coordinate fields. In Fig.3.3.12 on p. 991, four points are shown that define the posing of a linear dimension on a leader shelf. If manual text positioning is selected (see Table 3.3.3 on p. 977), the field p3 specifies also the location of the dimension text; if on-shelf location is selected, the field specifies the leader's start point. The p4 field is available in case one of the on-shelf location variants is selected (see Ibid.).





Tab. 3.3.1. Control elements on the **Dimension** tab

Element	Description
Type	The element specifying dimension line directions. Depending on the type of the dimension created, this element can have different appearance and contents (see Table 3.3.2).
Text	The field containing the dimension text. To edit the default text or input a new text, click in the field. Detailed information about dimension texts is presented in Section 3.3.1.3 on p. 979. Direct symbol input into the Text field is impossible.

Tab. 3.3.2. The **Type** element

Dimension Type	Contents of the Type Element
	Linear Dimension Horizontal
	
	
	Arc dimension Parallel Extension Lines
	Extension Lines from Center
	Level Dimension Front View or Cut
	Top View with Leader
	Top View Immediately in Drawing
	Diameter Dimension Full extension line
	
	Radial Dimension Radial Dimension Not from Circle Centerpoint

Tab. 3.3.2. The **Type** element

Dimension Type	Contents of the Type Element
	Radial Dimension from Circle Centerpoint
	Angular Dimension Minimum (acute) angle
	Maximum (obtuse) angle
	Greater than 180°

* Only for a simple linear dimension.

3.3.1.2. Options Settings

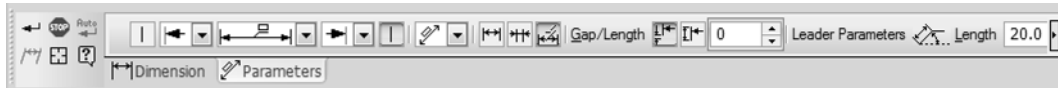




Fig. 3.3.3. The **Options** tab on the Property bar while specifying a linear dimension.

The dimension control elements on the **Options** tab are presented in Table 3.3.3.

Tab. 3.3.3. Control elements on the **Options** tab

Element	Description
	Extension line The radio button specifying the extension line visibility. Only for dimensions with extension lines. For radial dimensions and a linear dimension with a broken line, there is only one switch in the Parameters tab. For any other linear dimensions and for diameter dimension there are two such switches.
	Arrow The list serves to select the type of the dimension arrow. The list of arrows available for selection as well as the order of arrows in the list is defined by the setting of the arrows filter made for the current document (see Section 9.2.6.10.5 on p. 1774).

Tab. 3.3.3. Control elements on the **Options** tab

Element	Description
Text Location	This list serves to specify the desired way of positioning the dimension text: Auto , Manual , On Leader shelf Left , On Leader shelf Right , On Leader shelf Top , On Leader shelf Bottom . For any dimensions, except a linear with a common dimension line, angular with a common dimension line and height dimensions. For details on dimension text location control see Section 3.3.1.5 on p. 987.
Text Location	This list serves to select the choice of positioning a dimension text relative to the dimension line: Parallel, above line , Parallel, in the line break , Horizontally, in the line break . For any dimensions, except a linear with a common dimension line.
Placing arrows	The group of radio buttons for selection of arrow location variants relative to the dimension extension lines. <ul style="list-style-type: none"> ▼ If you need to have the arrows placed inside the range between the extension lines, activate the Inside switch. ▼ If you need to have the arrows placed outside the range between the extension lines, activate the Outside switch. ▼ If you would like the system pick the arrows placement option automatically, then activate the Auto switch.
Branchings	The group of radio button that allow to select the method to create branches: From Shelf Start , From Shelf End . Only for a simple radial dimension. The switches are available if one of the types of arrangement of dimension text on leader shelf has been selected from the Text Location list.
Gap/Length	The group of elements allows creation of dimension lines with gaps or fixed-length (see Section 3.3.1.8 on p. 989). For linear (except dimension with broken line), diameter and radial dimensions. Elements are available when extension lines display is enabled.

Tab. 3.3.3. Control elements on the **Options** tab

Element	Description
Leader Parameters	The group of fields serves to define a leader line parameters. For any dimensions, except a linear with a common dimension line. Creating a dimension with a text placed on a leader shelf, the length of the leader is displayed in the Length field, and the Angle field represents the slope angle between the leader and the main axis. You can input these values from the keyboard if the automatic object creation option is disabled (see Section 1.4.2.10 on p. 93) and only when the leader has been created (the shelf start point has been specified), editing of the created leader is actually performed in these fields.
Bend angle	The field for entering a number of dimension line bend angle. Only for a bent radial dimension.
Pointer from Text to Arc	The option to manage the drawing of the pointer from a dimension text to a measured arc. For arc dimensions only.
By default	If the option is enabled, all current settings on the Options tab will be used to create all the following dimensions of the given type until the end of the session. If the option is disabled, then the setting affects only the current dimension (the one being created).

3.3.1.3. Dimension Text input

Character input into a dimension text is performed in a special dialog (Fig. 3.3.4) called by clicking in the field **Text** on the **Dimension** tab.

The control elements of this dialog are represented in Table 3.3.4.

Inputting and editing a dimension text, you have to pay attention to the following:

- ▼ If the value range limits are included in the dimension text, and the class of accuracy is not, then the nominal value will not be shown in the dimension text.
- ▼ If the dimension value range limits display is enabled, but the class of accuracy is not defined, then changes in the dimension geometry (for example, when rebuilding an associative dimension) do not cause recalculation of the value range limits.

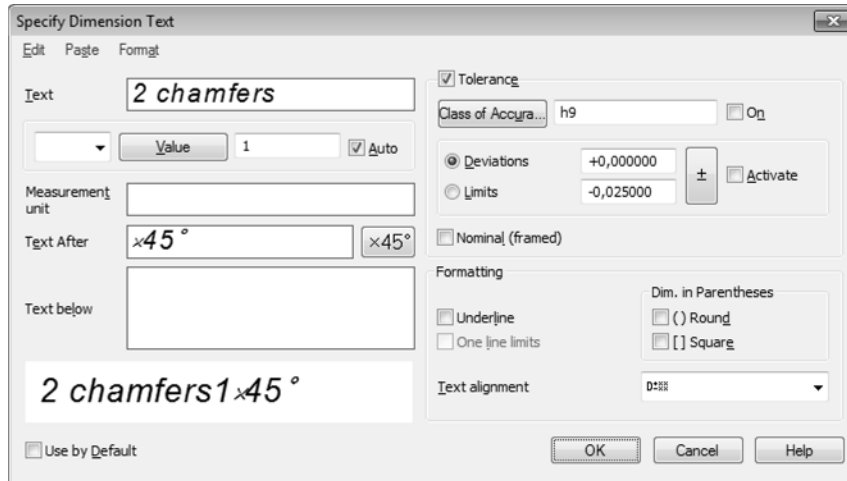


Fig. 3.3.4. Linear dimension text Input dialog

Tab. 3.3.4. Dimension text input dialog

Element	Description
Text Before*	The field for prefix input: text preceding the dimension value.
List of Symbols	This drop-down list makes it possible to choose the required symbol for setting in front of the dimension number. You can choose a symbol for diameter, square, radius, metric thread designation or the No Symbol option. If you need to enter another symbol, enter it in the Text before field using the Insert menu command.
Value	This is the field for displaying the dimension's value. You can input the value from the keyboard. Specifying any dimensions, except for angular ones, you can select the value from the user menu. To do so, click Value . The user menu will appear on the screen. As the default value, it contains normal linear dimensions by GOST 6636-69.
Auto	This option controls the dimension value specification method. If the option is enabled, the size value is defined automatically. After manual value entry (also from the user menu), the option will be automatically disabled. To restore the automatically specified value, enable the Auto option again. The precision of the automatically calculated value can be specified during the current document customization (see Section 9.2.6.10.12 on p. 1781). If a dimension value was entered manually, then the accuracy setting does not affect it.

Tab. 3.3.4. Dimension text input dialog

Element	Description
Display Format	The drop-down list allows to select a format of displaying the angle nominal value in the dimension text. These options are present in the dialog while specifying angle dimensions only. The Degrees, minutes, seconds option makes it possible to display the angle's value in the XX°XX'XX" format, while the Decimal system option enables the XX,XXX° format.
Unit*	The field for dimension unit designation. The text will be displayed in the dimension text string immediately following the tolerance. While specifying angle dimensions, the Unit field is absent.
Text After*	The field for suffix input: a text following the dimension value.
x45°	This button makes it possible to add the x45° line to the text following the dimension's value.
Text Under*	The field for text which will be positioned below the dimension text.
Preview Field	This field contains the dimension text as it will displayed in the document (except for the text below the dimension text). This field is not available for manual entry.
Use Default	If the option is enabled, all current settings will be saved for using during creation of the following dimensions of the type until the end of the working session. If the option is disabled, current settings are applied only to the current dimension (being created or edited), while new dimensions have the default setting (see Sections 9.2.6.10.10 on p. 1779 and 9.2.6.10.11 on p. 1780).
Tolerance	This group of elements makes it possible to assign a tolerance for the dimension being specified (see Table 3.3.5).
Layout	This group of elements makes it possible to adjust additional parameters of the dimension text's layout (see Table 3.3.6).

* Filling in the fields **Text Before**, **Unit**, **Text after**, **Text Below**, you can retrieve text from user menus. To call the user menus, double-click in the field you are filling in (do not confuse this action with the context menu also available in these fields). The default contents of the user menus in the specified fields is presented in Figure 3.3.5.

Tab. 3.3.5. Elements for assigning a tolerance to dimension

Element	Description
Tolerance	<p>This option makes it possible to control the dimension's tolerance setting. If the option is enabled, the current dimension can have a tolerance. At the same time, elements for tolerance setting are available in the dialog. If the option is disabled, these elements will not be available.</p> <p>Note that the Tolerance option is disabled, class of accuracy and deviations (limits) chosen for the dimension will be removed. Besides that, Enable and Nominal (in frame) options will be disabled (the options' description can be found below). Re-enabling the Tolerance option does not restore data related to the options and their status.</p>
Class of Accuracy	<p>This field displays the dimension's assigned class of accuracy. In order to select the required class of accuracy, click the Class of Accuracy button. For information about accuracy class selection, see Section 3.3.1.4 on p. 985.</p> <p>The Enable option on the right side from the Class of Accuracy field controls displaying the class of accuracy in the dimension text. If the option is enabled, the class of quality is displayed in the dimension text; if the option is disabled, it is not displayed.</p> <p>When assigning angle dimensions, elements for class of accuracy selection will not be available.</p>

Tab. 3.3.5. Elements for assigning a tolerance to dimension

Element	Description
Deviations Limits	<p>The group of elements makes it possible to add limit deviations or set limit values for a dimension text.</p> <p>Limit deviations are added, if the Deviations option is selected, while limit values for the dimension are assigned if the Limits option is selected.</p> <p>For an angular dimension, only manual entry of deviations and limit values is possible.</p> <p>For linear, radial, and diameter dimensions, deviations and limit values can be set manually or generated automatically, if a class of accuracy is selected for the dimension. Note that after manual entry of deviations or limit values, class of accuracy selection will be canceled (the Class of Accuracy field will become empty).</p> <p>When entering deviations, you have the \pm button available. You can use it to quickly make the deviations equal. After clicking it in the current deviation input field the value is passed to the second deviation field (the current is the field in which the cursor is located). In the upper field, the “+” sign is inserted before the value, while in the lower field— the “-” sign.</p> <p>The Enable option to the right from the \pm button controls displaying deviations or limit values in the dimension text. The option is available in the dialog for all dimensions, except for angular ones. If the option is enabled, deviation options (limit values) are displayed in the dimension text; if the option is disabled, they are not displayed.</p>
Nominal (framed)	<p>You can use this option to draw a frame around the part of the dimension text which includes the symbol, dimension's value, class of accuracy and deviations (limits). Once this option is enabled, the system will automatically stop displaying the class of accuracy and deviations (limits). If necessary, you can re-enable displaying them at any time.</p>

* If this option is not available, it means a class of accuracy number is set, for which the insertion in the dimension text is disabled. The accuracy class limiting number is specified during the current document customization (see Section 9.2.6.10.12 on p. 1781).



If variables of individual deviations are created for a dimension (see Section 7.1.5.2 on p. 1592) and the **Expression** cell of the Variables' Window (see Section 7.1.1.3 on p. 1571) contains at least one of them, the following tolerance assignment elements will not be available in the dimension text entry dialog: the **Class of Accuracy** button, the field for class of accuracy setting, the option controlling displaying of the class of accuracy, as well as the fields of limit deviations/values of the dimension.

Tab. 3.3.6. Layout setup elements

Element	Description
Underline	You can use this option to enable underlined display of the part of the dimension text which contains the symbol, dimension's value, class of accuracy and deviations (limits).
One line limits	You can use this option to display the dimension's limit values one by one separated by dash rather than one under another. This option is available if the Limits option is enabled.
Dimension in Parentheses	This group of options makes it possible to display in parentheses the part of the dimension text which contains the symbol, dimension's value, class of accuracy and deviation. Enable the option which corresponds to required type of brackets (round or square). The selected elements are taken into parentheses together with the frame and underlining, if there are any.
Text Alignment	This drop-down list makes it possible to select the method of positioning deviations or limit values of a dimension relative to its nominal value. Choose the required position option from the list.

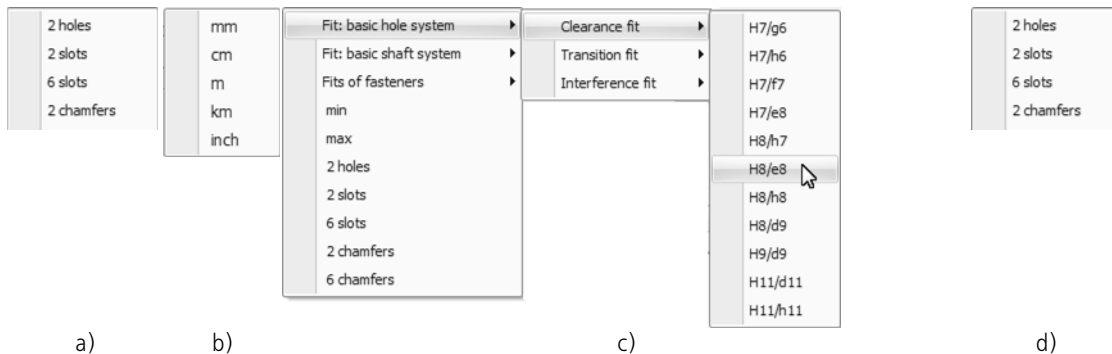


Fig. 3.3.5. User menu in the fields: a) **Text Before**, b) **Unit**, c) **Text After**, d) **Text below**

While filling in the **Text Before**, **Unit**, **Text After**, **Text below** fields, the main menu of the dialog is available. It contains commands text editing and formatting commands. These commands are used in the same way as when working in the text editor (see Section 4.1.2 on p. 1293).

After performing all necessary operations in the dialog, close it by clicking the **OK** button. The layout of the dimension text in the **Text** field of the **DIMENSION** tab in the Property bar will change in accordance with the adjustments made.

You can input the components of a dimension text using the **Text** field. To do that, call the context menu in this field (see Fig. 3.3.6) and select the command you need.

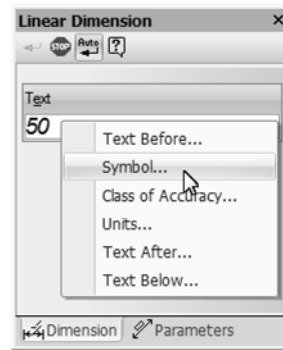


Fig. 3.3.6. Selecting a command from the context menu in the **Text** field.

The dialog for entering the selected part of the Dimension Text will be displayed.

In Fig.3.3.7 an example of such a dialog is shown. You can also call the user menu in it.

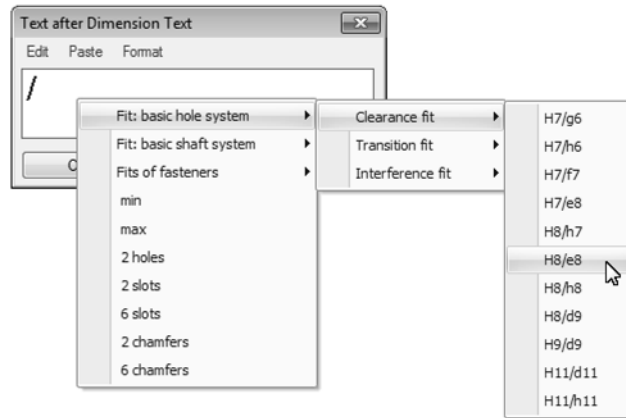


Fig. 3.3.7. Enter Dimension Text Suffix Dialog

3.3.1.4. Accuracy class selection

After clicking the **Class of Accuracy** button in the dimension text dialog, an accuracy class selection dialog will appear on the screen (Fig. 3.3.8).

In this dialog, you can specify the necessary accuracy class, as well as select it according to the extreme deviations. The control elements of this dialog are represented in Table 3.3.7.

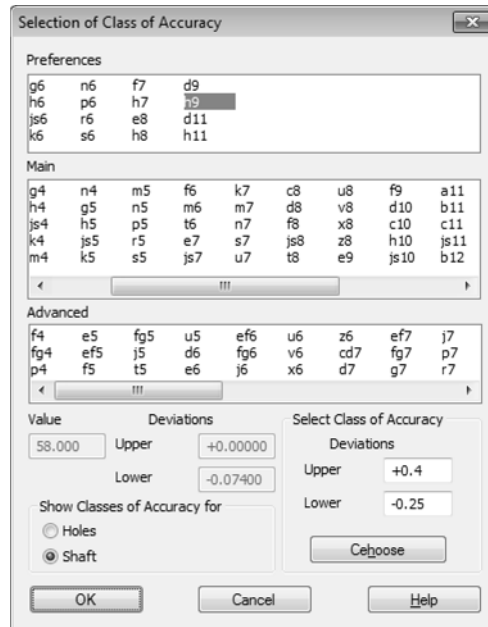


Fig. 3.3.8. Select Class of Accuracy dialog

Tab. 3.3.7. Control elements in the accuracy class selection dialog

Element	Description
Preferences	The fields containing the lists of preferred, main, and auxiliary accuracy classes corresponding to the selected system (shaft or opening). To select an accuracy class, click on it.
Main*	
Advanced *	
Value	The reference field containing the dimension nominal value.
Deviations	A group of reference fields containing the values of current dimension deviations. These deviations are calculated according to the accuracy class or are manually specified.
Show Classes of Accuracy for	This group of options make it possible to specify the system in which tolerance will be assigned (in the hole system or in the shaft system).
Select Class of Accuracy	This group of elements makes it possible to define class of accuracy based on assigned dimension's limit deviations. Enter the required values of dimension extreme deviations into the Upper and Bottom fields and click the Select button. Only those values that fit in the deviation range will remain in the list.

- * The presence of main and auxiliary accuracy classes in the dialog depends on the parameters of the new dimensions. To make one or both lists invisible, call the **Service — Parameters... — Current drawing/fragment — Dimensions — Tolerances and tolerance range — Parameters**. In the right part of the appearing dialog, disable the necessary options in the **Classes of Accuracy** group.

3.3.1.5. Dimension lines and dimension text management

This section contains descriptions of methods available for management of the dimension line and text of any size except for linear dimension with a common dimension line, angular dimension with a common dimension line and height dimension.

Variants of positioning the dimension text are in the **Text Location** list on the **Parameters** tab of the Properties bar (see Fig.3.3.3 on p. 977). In the list, select the desired item. Therewith, the initial dimension text position will correspond to the selected variant.

When marking dimensions you can modify the position of the dimension text in one of the following ways.

- ▼ Selection of another variant of positioning the dimension text from the list.
- ▼ Moving the cursor rightward, leftward, upward or downward.

If an automatic or manual dimension text placement is selected, then as the cursor moves rightward/leftward switching between these variants is done. The dimension text is then positioned so that it does not intersect with extension lines and does not appear to be under arrows. To position the text over the dimension line at your discretion, move the cursor while pressing the **<Alt>** button.

If a variant of the dimension line positioning on the shelf is selected and the dimension position is fixed, moving the cursor to the right/left (for a horizontal shelf) or up/down changes the shelf's direction to the opposite.

- ▼ Using the **<Ctrl>** button and the mouse wheel. When creating a linear dimension, it is available for the first chain dimension only.

If an automatic or manual dimension text placement is selected, then after moving cursor with **<Ctrl>** key pressed the dimension position is fixed, and the dimension obtains a leader line and a horizontal flange. Pay attention to the fact that in this case the start point of leader line is the midpoint of the dimension line. By moving the cursor, you can specify the flange start and direction— right or left.



While specifying diameter and radial dimensions, pressing and holding the **<Ctrl>** key does not fix position of dimension.

To rotate the shelf, keep the **<Ctrl>** button pressed and spin the mouse wheel in both directions. If you spin the mouse wheel backward, the shelf will be rotated clockwise. If you spin the mouse wheel forward, the shelf will be rotated counterclockwise.

- ▼ Enabling/Disabling dimension text on leader shelf with the **Dimension of a Shelf** context menu command.

3.3.1.6. Setting up the Dimensions in Current and New Documents

Formatting of a drawing can be much faster if the main parameter set is used as default — creating a new dimension, only small corrections will be needed.

KOMPAS-3D uses such default symbol settings that are most commonly used in design documentation. If necessary, you can modify the dimensions options. Dimension setting options are described in Section 9.2.6.10 on p. 1771. The setting will be saved in the current document and will not change when opening it on another workplace.



If changing the dimension settings in the current document is not desirable, you can set up a parameter combination that will be used as default on dimensioning until the end of the current session.

To do that, while specifying the first dimension value of each type (linear, angular etc.), set it up as desired and enable the **Default** option on the **Options** tab of the Property bar and in the dimension value input dialog.

3.3.1.7. Dimension Line Alignment

After specifying linear and angular dimensions in the drawing, you can run automatic alignment of dimension lines by sample. As a sample, one of the dimensions in the drawing must be selected.

After the alignment, the length of the extension lines is changed to place the linear dimension lines in line, and the angular dimension lines on the same circle (or on circles with the same radius).

The dimensions that need to be aligned can be specified after (Method 1) or before (Method 2) calling the command.

To align dimensions, do the following:

Method 1.



1. Call the command **Align Dimension Lines** from the **Tools** menu. The command is available if a graphic document has at least one linear or angular dimension.
2. Move the cursor to the sample dimension whose dimension line is used to perform alignment and click the left mouse button on it. The sample will be highlighted.
3. Point the cursor sequentially to one or more dimensions for alignment. When you move the pointer over a dimension that can be aligned, it will be highlighted. After clicking on it, its dimension and extension lines will be rearranged.



A dimension can be aligned if it corresponds to the type of the sample dimension. For linear dimensions, it is also necessary that their dimension lines be parallel to the dimension line of the sample.

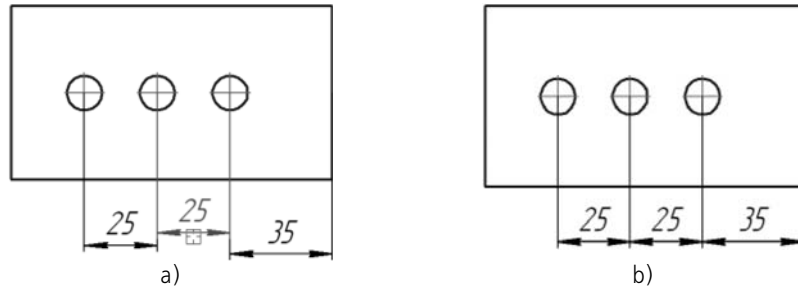


Fig. 3.3.9. Linear dimension alignment example:
a) specifying sample, b) alignment result

The sample dimension designation remains highlighted as the command is executed.



To start aligning another group of designations, click the **Select Other** button on the Advanced Control panel. Specify the sample and the designations to be aligned.

Method 2.

1. Select the dimensions on the drawing to align. If certain objects, other than dimensions, are selected, they will be ignored as the command is executed.
2. Call the **Align Dimension Lines** command and specify the sample dimension. Those dimensions that can be aligned based on the sample, will be rearranged; the rest will be ignored.



To align several groups of dimensions at a time, for example, vertical and horizontal dimensions, select all the dimensions that can be aligned (use method 1 or method 2) and sequentially specify the samples required for them with the **Select Other** button.

3.3.1.8. Creating a Gap between the Extension Line and the Snap Point

Sometimes, when creating a dimension, you may need to have the extension line drawn at some distance from the snapping point. It is possible when specifying linear dimensions of all types (except dimension with broken line), diameter and radial dimensions.

To implement this method, the group of control elements named **Gap/Length** is used. It is located on the **Options** tab of the Property Bar. Elements of the group are available when extension lines display is enabled (see Table 3.3.3 on p. 977).

You can fix both the length of extension lines and the gap between the reference point and the start of the extension line. To do this, enter the desired value in the field **Gap/Length**.



If you need to set the gap between the start of the extension line and the dimension snap point to a specified value, activate the **Gap of Extension Lines** switch. In this case, the extension line length will be adjusting as the dimension line is moved, while the gap remains constant and equal to the specified value.



If the specified value should define the length of the extension line, activate the **Extension Lines Length** switch. In this case, the gap will be adjusting as the dimension line is moved, while the extension line length remains constant and equal to the specified value.

If the value in the **Gap or Length** field is equal to zero, then the extension lines begin at the dimension snap points and can have an arbitrary length.

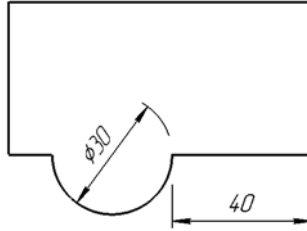


Fig. 3.3.10. An example of linear and diameter dimensioning with a gap

3.3.2. Linear Dimensions

The system allows you to specify linear dimensions using different methods. Most parameters are the same for different dimension methods. The difference is in the order used to specify defining points and measured objects.

3.3.2.1. Simple linear dimension



To specify a linear dimension, call the **Linear Dimension** command.

Select with the cursor the dimension snap points— **p1** and **p2** (the points of origin for the extension lines). For this purpose, specify the required points and/or objects in the document (see Section 3.3.2.1.2 on p. 991).

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Then specify the point determining the position of the dimension line **p3**.

If a manual dimension text placement is selected when setting the dimension display parameters, its position is also defined by the **p3** point (Fig. 3.3.11).

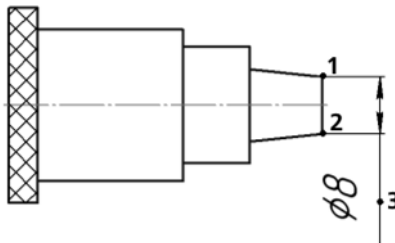


Fig. 3.3.11. Linear dimension with manual dimension text location

If on-shelf location of a dimension text is selected, then the point **p3** also defines the beginning of the leader. In this case, to specify the text location, you need to specify the starting point of the shelf — **p4** (Fig. 3.3.12).

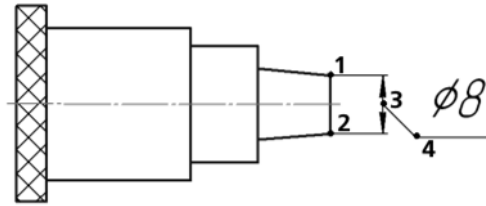
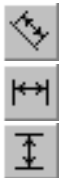


Fig. 3.3.12. Linear dimension on leader shelf

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

3.3.2.1.1. Dimension Orientation Management



By default, a dimension line is parallel to the line going through the dimension snap points. On the **DIMENSION** tab of the Property bar, the **Parallel to Object** switch is active.

To create a horizontal or vertical dimension, activate the respective switch.

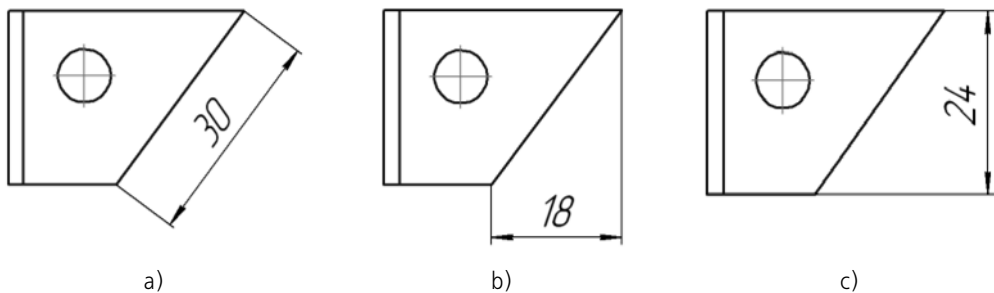
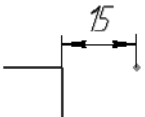


Fig. 3.3.13. Linear dimension orientation:
a) parallel to object, b) horizontal, c) vertical

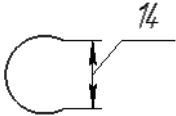
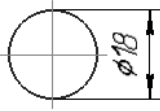
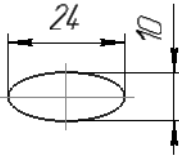

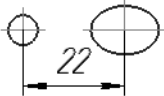
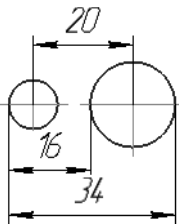
3.3.2.1.2. Specification of objects for dimension setting

In order to set dimension snap points, various objects can be specified in different combinations. All available combinations can be found in Table 3.3.8.

Tab. 3.3.8. Methods of setting linear dimension snap points

Specified objects	Resulting dimension	Comment
Two points		Can be specified both as individual points or as object handles.

Tab. 3.3.8. Methods of setting linear dimension snap points

Specified objects	Resulting dimension	Comment
Segment or arc (circular, ellipse arc)		Snap points are the extreme points of the specified object.
Circle		The diameter symbol is automatically added to the dimension text. In order to choose the dimension's position (horizontal or vertical), move the cursor.
Ellipse		A dimension for a semiaxis is created. In order to select the semiaxis for dimensioning, move the cursor.
Centerline (or a segment of the centerline style) and point		The dimension value is equal to twice the distance between the centerline and the point. The specified point and the point symmetrical to it with respect to the centerline become snap points of the dimension.
Circle and ellipse*		The dimension between the circle center and the ellipse center is created.
Two circles*		Depending on points in which circles were specified, the dimension is created between their centers or the dimension with extension lines tangent to circles.

* Arcs can also be used.

3.3.2.1.3. Linear Dimension with Slanted Extension Lines

Sometimes, it is necessary to slant the extension lines of a linear dimension, for example, as shown in Fig.3.3.14.

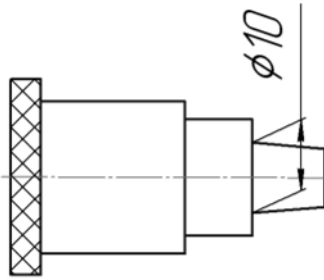


Fig. 3.3.14. Example of creating a dimension with slanted extension lines



A dimension of this type is formed based on a simple dimension with the help of the **Slant Dimension** button which is located on the Advanced Control panel.

This button is available if the following conditions are satisfied:

- ▼ automatic object creation is disabled (see Section 1.4.2.8 on p. 91),
- ▼ all defining points are specified (the dimension is completely defined),
- ▼ the dimension is parallel to the measured object (see Section 3.3.2.1.1 on p. 991).

After clicking the mentioned point, all options on the **Dimension** tab are replaced with one field named **Angle**. Enter in it the value of the inclination angle of the dimension extension lines. You can also “skew” a dimension with the mouse by dragging one of the appearing on the screen defining points.

After setting the desired position of the extension lines, release the **Slant a Dimension** button. On the **Dimension** tab, the regular control widget appear again.

3.3.2.2. Linear Dimensions with Broken Line



In order to specify a broken linear dimension, call the **Broken linear dimension command**.

Specify the segment for which you need to specify a dimension with broken line.

- ▼ If the image has a symmetry axis of the element being dimensioned, select it (the axis line must be parallel to the segment being dimensioned). The dimension is calculated as a doubled distance between the segment and axis. If necessary, you can edit the dimension text (see Section 3.3.1.3 on p. 979).
- ▼ If there is no axis of the dimensioned element, enter the dimension text manually (see Section 3.3.1.3 on p. 979).

Then specify the **p3** point determining the position of the dimension line and its length.

If on-shelf location of the dimension text is selected when setting up the dimension display parameters (see Section 3.3.1.2 on p. 977), specify the starting point of the shelf — **p4**. The start point of the leader is located in the middle of the dimension line. Its position cannot be changed.

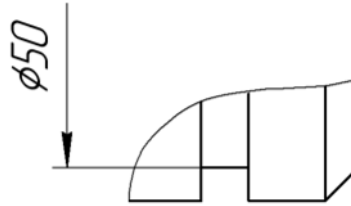


Fig. 3.3.15. Example of dimensions with broken line

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

3.3.2.3. Linear Dimension from Line to Point



To specify a linear dimension between two geometry elements, a segment and an instant point (including a defining point of another graphic object), call the **Linear Dimension from Line to Point** command.

Specify a line segment with the cursor, from which the dimension should originate.

The dimension extension lines will be parallel to that segment, while one of its ends will serve as the first snap point of the dimension.

Specify the second snap point of the dimension — **p2**. To do so, specify one of the following objects:

- ▼ arbitrary point;
- ▼ segment parallel to the dimension segment — point **p2** will become one of the segment;
- ▼ circle (arc of a circle) — point **p2** will become the tangent point of the circle and a straight line parallel to the dimension segment.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Specify the **p3** point determining the position of the dimension line and text.

The end of the line segment, to which the dimension line will be the closest, will be accepted as the first snap point of the **p1** dimension.

If on-shelf location of the dimension text is selected when setting up the dimension display parameters, specify the starting point of the shelf — **p4**.

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

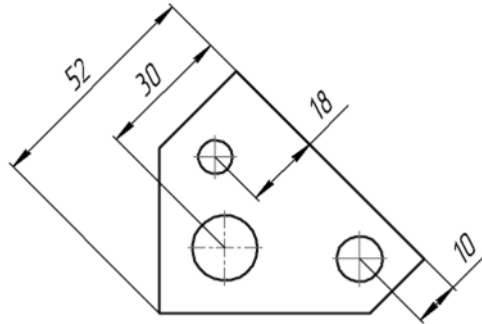


Fig. 3.3.16. Example of dimensioning between a line segment and the point

3.3.2.4. Linear Dimension from Common Datum



To make a group of linear dimension with a common datum, call the **Linear Dimensions from Common Datum** command.

Select the first snap point **p1**. It will be common for the group of dimensions being created.

Select the second snap point **p2** for the first group dimension.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Define the point determining the position of the dimension line **p3**.

If on-shelf location of the dimension text is selected when setting up the dimension display parameters, specify the starting point of the shelf — **p4**.

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

Specify sequentially the points (**p2**, **p3** and **p4**) for the other dimensions in the group.



On the **Dimension** tab of the Property bar, there is a group of switches named **Type**. They allow you to specify the orientation of each of the dimensions created (vertical or horizontal).



A group of dimensions created by the **Linear Dim from a Common Base** command is not an integral object— those are several simple linear dimensions whose first snap points coincide. Therefore, to use the same parameters for all dimensions in a group (for example, a text on a bent leader shelf or arrows of a certain type), after setting up the first dimension enable the option **By Default**.

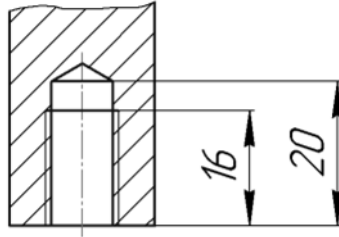


Fig. 3.3.17. Angular dimensions from a common datum creation

To proceed to creating another dimension group from a different baseline (datum), unfix the first base point (the field **p1** on the **DIMENSION** tab of the Property bar) and specify its new position.

3.3.2.5. Linear Chain Dimension



To construct a chain of linear dimensions, call the **Linear Chain Dimension** command.

Specify the first snap point of the dimension — **p1**.

Specify the second snap point of the dimension — **p2**.



On the **DIMENSION** tab of the Property bar, there is a group of switches named **Type**. They allow you to specify the orientation of each of the dimensions in the chain (vertical or horizontal).



You can choose an orientation only before fixing the first dimension in a chain.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Define the point determining the position of the dimension line **p3**. This position will be the same for all dimensions in the chain.

If on-shelf location of the dimension text is selected when setting up the dimension display parameters, specify the starting point of the shelf — **p4**.

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

Specify sequentially the points (**p2** and **p4**) for the rest of the dimensions in the chain.



A group of dimensions created by the **Linear Chain Dimensions** command is not an integral object—those are several simple linear dimensions created according to certain rules (the first snap point of each subsequent dimension coincides with the second snap point of the previous one; all dimension lines are located on the same straight line). Therefore, to use the same parameters for all dimensions in a chain (for example, a text on a bent leader shelf or arrows of a certain type), after setting up the first dimension enable the option **By Default**.

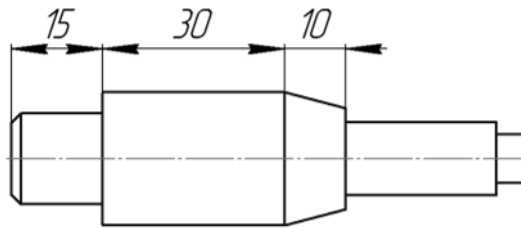


Fig. 3.3.18. Chain of angular dimensions creation

To specify the next chain dimension, unlock the first snap point of the chain (the **p1** field on the **Dimension** tab of the Property bar) and specify its new location.

3.3.2.6. Linear Dimension with Common Dimension Line



To make a group linear chain dimensions, call the **Linear Dimension with Common Dimension Line** command.

Select the first snap point **p1** for the group dimension.

Then select the second snap point **p2**.

On the **Dimension** tab of the Property Bar, there is the **Type** group of switches, that can be used to set the orientation of a group dimension (vertical or horizontal).



You can choose an orientation only before creating the first dimension in the group.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Define the point determining the position of the dimension line **p3**. This position will be the same for all dimensions in the group.

Then select the **p2** points for the rest of dimensions in a group.



A group of dimensions created by the **Linear Dimension with Common Dimension Line** command is not an integral object: it is a group of linear dimensions, whose first snap points coincide and the dimension text is positioned in a special way. Therefore, to use the same parameters for all dimensions in a group (for example, arrows of a certain type), after setting up the first dimension enable the option **By Default**.

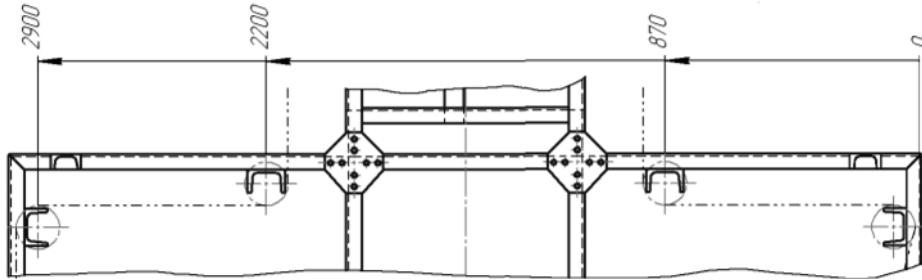


Fig. 3.3.19. Angular dimensions with a common dimension line creation

To specify the next group of linear chain dimensions, unlock the first snap point of the group (the **p1** field on the **Dimension** tab of the Property bar) and specify its new location.

3.3.2.7. Arc Dimension



To construct the dimension that indicates the length of an arc, call the **Arc Dimension** command.

Select with the cursor the arc that needs to be dimensioned.



On the **Dimension** tab of the Property bar there is the **Type** group of switches. Using them, you can define the direction of the extension lines: either from the center or parallel to the arc bisector radius the middle of the arc. If the arc included angle greater than 180 degrees, then you can only create a dimension with the extension lines from the center.

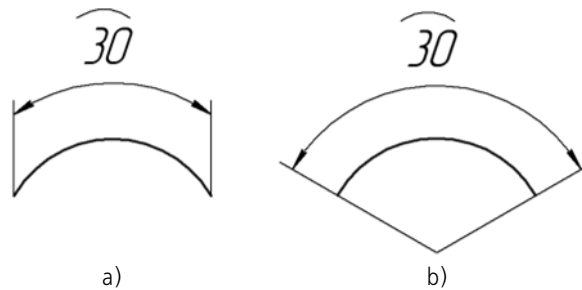


Fig. 3.3.20. Arc dimension: a) with parallel extension lines, b) with extension lines from the center

Then specify the point determining the position of the dimension line **p3**.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977). An arc sign is automatically placed above the dimension value.

On the **Parameters** tab of the Property bar, there is an option **Pointer from Text to Arc**. Enable this option to connect the arc and the text of its respective dimension by a leader. This may be needed, for example, when dimensioning concentric arcs with the same included angle (angular length) and the start angle.

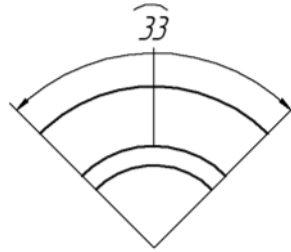


Fig. 3.3.21. Example of creating an arc dimension with a pointer from the text to the arc

If on-shelf location of the dimension text is selected when setting up the dimension display parameters, specify the starting point of the shelf — **p4**.

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

3.3.3. Diameter and Radial Dimensions

3.3.3.1. Diameter Dimension



To specify a diameter dimension, call the **Diameter Dimension** command.

Specify the circle to measure.



The dimension line can be Full or 'with Broken Line'. To select the appropriate variant, use the group of switches named **Type** on the **Dimension** tab of the Property bar.



If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

If an automatic or manual dimension text placement is selected when setting the dimension display parameters, select the **p1** point defining the dimension line and text position (Fig. 3.3.22).

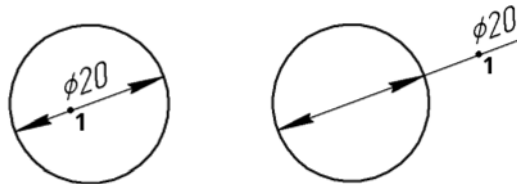


Fig. 3.3.22. Diameter dimensions with automatically placed text

If on-shelf location of the dimension text is selected, then the text may be positioned as shown on Fig. 3.3.23.

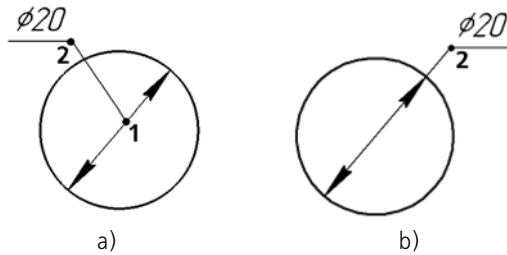


Fig. 3.3.23. Diameter dimensions with text on a shelf:
 a) leader line does not coincide with dimension line,
 b) leader line coincides with dimension line

- ▼ To position dimension text as shown on Fig.3.3.23,a, specify point **p1**, defining the beginning of leader line, and point **p2**— the shelf start point. Point **p1** must be located inside a circle.
- ▼ To position dimension text as shown on Fig.3.3.23,b, specify the shelf start point **p2** only. Specified point must be located outside a circle.

Methods to control the dimension line and location of dimension text are described in Section3.3.1.5 on p. 987.



A broken dimension line projects (extends) beyond the circle center by the distance equal to 1/5 of the radius, but not less than by the distance set in this document for the dimension line projection (extension) beyond the text. This value is specified while setting up the current document (see Section 9.2.6.10.2 on p. 1772).

3.3.3.2. Simple Radial Dimension



To construct radial dimension, call the **Radial Dimension** command.

Select with the cursor an arc or circle that needs to be dimensioned.

If needed, edit the dimension text (see Section3.3.1.3 on p. 979) and set up the dimension display parameters (see Section3.3.1.2 on p. 977).

If an automatic or manual dimension text placement is selected when setting the dimension display parameters, select the **p1** point defining the dimension line position (Fig.3.3.24).

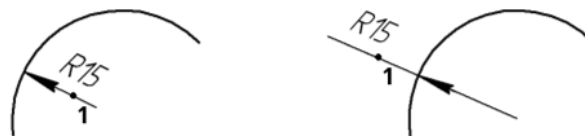


Fig. 3.3.24. A radial dimension with an automatically positioned dimension text

If the dimension text is to be positioned on a flange, specify the flange start point **p2** (Fig.3.3.25). To create dimensions of identical radii on the common shelf, shelves with several branches are used (see Section3.3.3.2.1).

Methods to control the dimension line and location of dimension text are described in Section3.3.1.5 on p. 987.

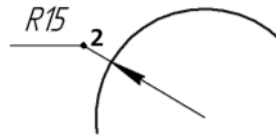


Fig. 3.3.25. Radial dimension on a shelf

A radial dimension can be specified either from a circle center or not. In both cases, the dimension line belongs to a line going through the center of the circle being dimensioned. The difference is the following. If a dimension is created from a center, then its dimension line length cannot be less than the radius (Fig.3.3.24, 3.3.25). If a dimension is created not from a center, then its dimension line can be of an arbitrary length (Fig.3.3.26).

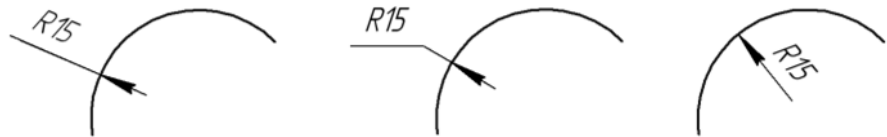


Fig. 3.3.26. Radial dimension not from circle centerpoint



To select an appropriate variant, use the **Type** switch group on the **Dimension** tab of the Property bar.

3.3.3.2.1. Radial size on the shelf with several branches

In accordance to GOST, dimensions of identical radii can only be specified on the common shelf. Shelves with several branches are used for this type of dimension specification (Fig.3.3.27).

Fig. 3.3.27. Radial dimension of the shelf with two branches
a) From Shelf Start, b) From Shelf End.

Specification of dimension on the common shelf is only possible for non-concentric circles (circular arc) with equal radii.

To create dimension for two or more circles (circular arcs) complete the following steps.



1. Call the **Radial dimension** command.
2. Select with the cursor the first circle (circular arc) that needs to be dimensioned. The dimension ghost will be displayed.

3. Branches of dimension (dimension lines to other circles and circle arcs) may be created starting from a shelf's beginning or end.

- ▼ In order the next branch of the dimension to begin from the shelf's start (Fig. 3.3.27), a), specify the second circle (circle arc). The shelf will be created automatically. The ghost of the dimension will change— it will consist of two dimension lines and a shelf.
- ▼ In order the next branch of the dimension to begin from the shelf's end (Fig. 3.3.27), b), it should be created in advance.

In order to create a shelf, position the cursor keeping the *<Ctrl>* key pressed or open the **Parameters** tab of the Property bar and select one of the variants of positioning the title block text on the shelf from the **Text positioning** list. The ghost of the dimension will change— it will consist of dimension line and shelf.



The **Branches** group of switches will become available. You will be able to choose the method of branch construction — **From the shelf's start** or **From the shelf's end**. Enable the **From the shelf's end** switch. A branch will be added to the ghost's dimensions, starting from the shelf's end.

4. Specify other circles (circular arcs) that need to be dimensioned.

You can create any number of branches by selecting circles and arcs of the same radius. The branch creation variant (from the beginning of the shelf or from its end) depends on which switch of the **Branch** group is enabled.

5. After specifying all necessary circles (circles' arcs), finish assigning the dimension by specifying the **t2** point of the shelf's beginning (see Fig. 3.3.27).

Henceforth, you will be able to change the parameters of the created dimensions, edit its handles, add branches (see Section 3.3.3.2.2). If you need to make changes during creation of a dimension, disable smart creation (autocreation) of objects.

3.3.3.2.2. Radial dimension editing

Radial dimension can be edited by changing its parameters in the Property bar (see Section 3.3.1.2 on p. 977).

If a radial dimension on shelf is selected for editing or, when changing the dimension parameters, its dimension text is set to be positioned on a shelf, you can add branches of the dimension. To switch to a dash adding mode, press the **Add branches** button on the Special Control Panel. Branches are added automatically when a circle (circle arc) is specified, and the circle's radius is the same as the radius of the circle to which the edited dimension is specified.



You can delete branches or modify the shelf's position in the handle editing mode. This mode is available, if dimensions have branches. To enter the mode, click the **Edit points** button on the Special Control Panel.

The following actions can also be performed in the handle editing mode:

- ▼ Move the shelf by changing the coordinates of the **p2** point (see Fig. 3.3.27) or by “dragging” it with mouse to the desired place. When defining the position of point **t2**, you can use snaps (see Section 3.1.1 on p. 865) and geometrical calculator (see Section 3.1.2 on p. 871).
- ▼ Rotate the shelf by rotating the point at the shelf's end around **t2** point with a step of 90°.
- ▼ Delete a branching by selecting branching end defining point and pressing the *<Delete>* key.

Note that the modes for adding branches or handle editing cannot be enabled at the same time. If one of these modes is enabled, the other one is disabled automatically.

3.3.3.3. Bent Radial Dimension

The bent radial dimension is used when you need to dimension an arc of a very small curvature, for example, as shown in the Figure3.3.28.

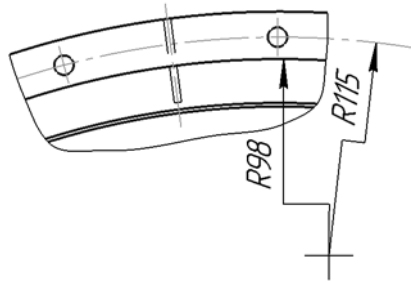


Fig. 3.3.28. Bent radial dimensions

In this case, the dimension line appears as a polyline, where its link, which is terminated with the dimension arrow, coincides with the actual radius in the direction of the dimension snap point on the arc.



To specify a bent radial dimension, call **Bent Radial Dimension** command.

Select with the cursor an arc or circle that needs to be dimensioned.

If needed, edit the dimension text (see Section3.3.1.3 on p. 979) and set up the dimension display parameters (see Section3.3.1.2 on p. 977).

If an automatic or manual dimension text placement is selected when setting the dimension display parameters, select the **p1** point defining the dimension line and text position (Fig.3.3.29).



Fig. 3.3.29. Bent radial dimensions with an automatically positioned dimension text

If the dimension text is to be positioned on a flange, specify the flange start point **p2** (Fig.3.3.30).

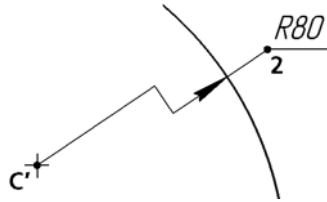


Fig. 3.3.30. Radial dimensions with text on a shelf

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

Specify the location of the fake circle center (in Figures it is denoted with the point C'), that is located closer to the arc than the actual center.



Bent dimensions specified one after another to concentric arcs or circles are automatically created with the same fake center—a point that had been specified for the first one of them (see Fig. 3.3.28).

3.3.3.3.1. Editing dimension line with a bend using mouse

You can edit dimension line with a bend changing the location of its handles with the mouse in the document window.

More information about editing by changing the location of its handles is given in Section 3.4.1.1.3 on p. 1114.

Dimension line with a bend has following handles:

- 1 — a point to change the location of the fake center,
- 2 — a point for bent offset along the dimension line,
- 3 — a point for changing the line slope angle; for the dimension line, if its text is positioned manually or on the shelf (see Table 3.3.3 on p. 977), point 2 also makes it possible to move the text or shelf,
- 4 — a point to change the shelf direction.

The number and location of handles depends on the method of positioning the inscription (Fig. 3.3.31).

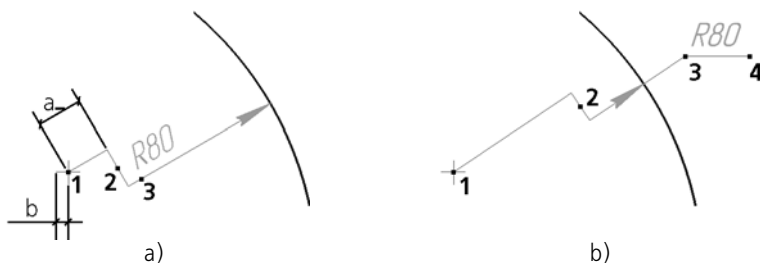


Fig. 3.3.31. Handles of the dimension with text positioned:
a) manually, b) on shelf

It should be noted that when moving a bend point the length of the line between the center and the bend is limited by the fake center identifier, i.e. $a > b$ (see Fig.3.3.31, a).

To change the default center designation dimensions in the current document, call the following command: **Service — Parameters... — Current drawing/fragment — Lines — Axis line.**

3.3.4. Angular Dimensions

3.3.4.1. General Angular Dimension Handling Techniques

3.3.4.1.1. Specification of Angle Sides

To specify angular dimensions of all types, you can specify linear objects or points which, if connected, form sides of the angle.

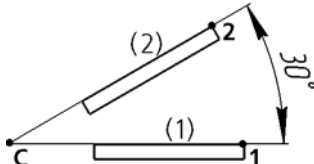
As a straight object (hereinafter - the segment), you can use:

- ▼ a segment, polyline link, polygon side,
- ▼ object of the **centerline** type,
- ▼ auxiliary straight line,
- ▼ axis of the absolute coordinate system or view coordinate system.

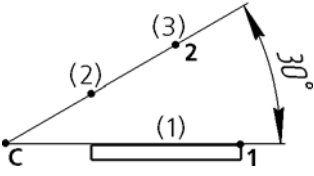
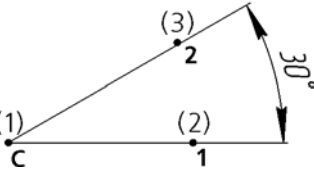
Points can be specified in the document, set by entering the coordinates on the property bar or using the geometric calculator (see Section 3.1.2 on p. 871).

Methods of specifying angle sides are given in Table3.3.9.

Tab. 3.3.9. Methods of specifying angle sides when creating an angular dimensions

Method*	Description
By two line segments	
	<ul style="list-style-type: none"> ▼ This first and the second side of the angle are set by specification of segments. One of the end points of the first segment will be accepted as the dimension's snap point t1; for the second segment, snap point t2 . <p>After specifying the sides, angle vertex position is computed automatically.</p>

Tab. 3.3.9. Methods of specifying angle sides when creating an angular dimensions

Method*	Description
<p>By a line segment and two points</p> 	<ul style="list-style-type: none"> ▼ The first side of the angle is specified by specification of a segment. One of its endpoints will be accepted as the first snap point of the t1** dimension. ▼ The second side of the angle is specified by specification of two points. Line segment and straight line passing through the points form an angle. One of its points will be the second p2 dimension snap point. <p>After specifying the sides, angle vertex position is computed automatically.</p>
<p>By three Points</p> 	<ul style="list-style-type: none"> ▼ The first side of the angle is specified by specification of the angle vertex and a point. ▼ The second side of the angle is specified by specification of a point. Straight lines passing through the vertex and each of points form an angle. <p>The points are the binding points of the created dimensions t1 and t2.</p>

* The sequence of selecting objects in Figures is marked as numbers in brackets.

** If an auxiliary straight line or an axis of the coordinate system is specified as the angle side, the dimension's snap point corresponding to the side will coincide with the angle vertex.

To create an angular dimension with a broken line, the symmetry axis is specified similarly instead of the second side.

3.3.4.1.2. Managing Dimension Orientation



The orientation of a newly created angular dimension is automatically defined by the system.

If the angle is acute, the switch **Acute Angle** in the **Type** switch group on the **Dimensions** tab will be active. If the angle is obtuse, the **Obtuse Angle** switch will be active. In Fig.3.3.32, possible locations of selecting the angle sides and the respective automatically determined angles are shown.

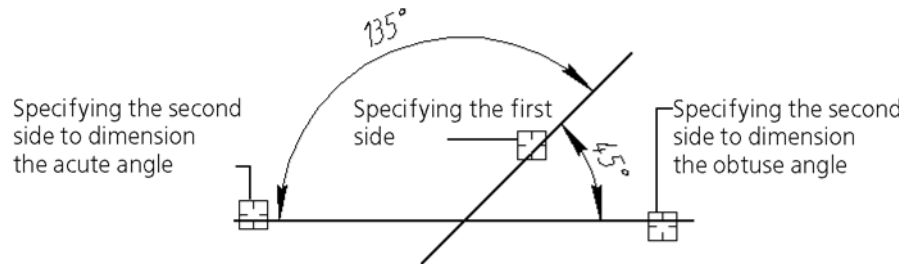


Fig. 3.3.32. Automatic detection of angular dimension type



If necessary, you can modify the system-proposed placement type by using the described switches, including bend creation of an angle greater than 180 degrees (such a configuration cannot be created automatically).

Note that there are angles that have greater than 180 degrees between two sides:

- ▼ a complementary angle adding to the acute angle between the sides to make up combined 360 degrees,
- ▼ a complementary angle adding to the obtuse angle between the sides to make up combined 360 degrees.

The selection of the desired variant is performed by specifying the **p3** point (Fig. 3.3.33).

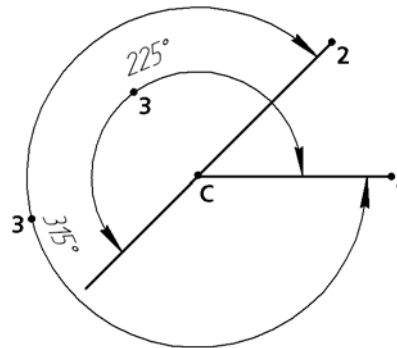


Fig. 3.3.33. Dimensioning an angle greater than 180° degrees between line segments



Changing the orientation of angular chain dimensions is only possible when auto creation is disabled (see Section 1.4.2.10 on p. 93).

3.3.4.1.3. Extension Lines View



Extension lines can be drawn from the center or otherwise, from binding points **t1** and **t2** (see Fig. 3.3.34).



To select an appropriate variant, use the **Extension Lines** switch group or call the **Extension Lines from Center** command from the context menu.

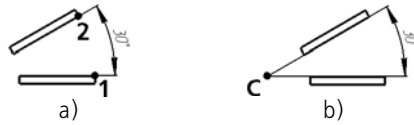


Fig. 3.3.34. Extension Lines View:
a) Not from Center, b) from Center

3.3.4.2. Simple Angular Dimension



To specify a simple angular dimension, call the **Angular Dimension** command.

Specify the first angle side.

After that, specify the second angle side (see Section 3.3.4.1.1 on p. 1005).

The angle vertex and snap points **p1** and **p2** coordinates appear on the Property bar.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

The **p3** point determines the position of the dimension line and text. The ends of the segments, to which the dimension line will be closest, will be accepted as the snap points of the dimension.

If a manual dimension text placement is selected when setting the dimension display parameters, its position is also defined by the **p3** point (Fig. 3.3.35).

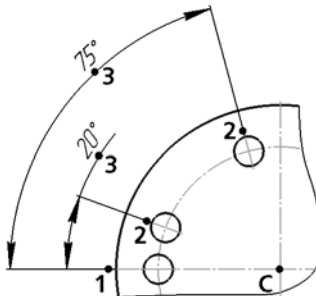


Fig. 3.3.35. Angular dimensions with manual location of dimension text;
Method of specifying angle sides— **by two line segments**

If on-shelf location of a dimension text is selected, then the point **p3** also defines the beginning of the leader. In this case, to specify the text location, you need to specify the starting point of the shelf — **p4** (Fig. 3.3.36).

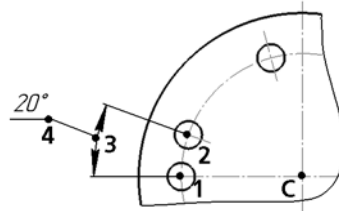


Fig. 3.3.36. Angular dimension on shelf;
Method of specifying angle sides— **by three points**

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

3.3.4.3. Angular Dimension from Common Datum



To create a group of angular dimension from a common base, call the **Angular Dimension from Common Datum** command.

Specify the first angle side common for the group of the dimensions being created.

After that, specify the second angle side for the first dimension of the group (see Section 3.3.4.1.1 on p. 1005).

The angle vertex and snap points **p1** and **p2** coordinates appear on the Property bar.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Define the point determining the position of the dimension line **p3**.

If on-shelf location of the dimension text is selected when setting up the dimension display parameters, specify the starting point of the shelf — **p4**.

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

Sequentially, select the second angle sides and the point **p3** (and **p4**) for the rest of the dimensions in the group.



A group of dimensions created by the **Angular from a Common Base** command is not an integral object— those are several simple angular dimensions whose first sides coincide. Therefore, to use the same parameters for all dimensions in a group (for example, a text on a bent leader shelf or arrows of a certain type), after setting up the first dimension, enable the option **By Default**.

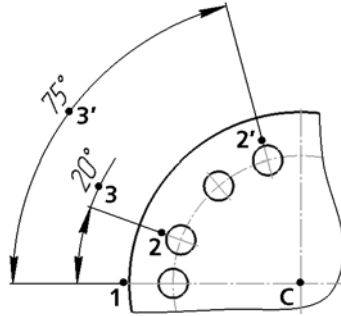


Fig. 3.3.37. Angular dimensioning from a common base;
Method of specifying angle sides— **by two line segments**



To specify a group of dimensions from another datum, click the **Reselect** button and specify a new angle side.

3.3.4.4. Chain Angular Dimension



To create a chain of angular dimensions, call **Angular Dimension Chain** command.
Specify the first angle side.

After that, specify the second angle side (see Section 3.3.4.1.1 on p. 1005).

The angle vertex and snap points **p1** and **p2** coordinates appear on the Property bar.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Define the point determining the position of the dimension line **p3**. This position will be the same for all dimensions in the chain.

If on-shelf location of the dimension text is selected when setting up the dimension display parameters, specify the starting point of the shelf — **p4**.

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

Sequentially, select the second angle sides — segments, point **p3** (and **p4**) for the rest of the dimensions in the chain.



All segments selected to create a chain angular dimension have to pass through one point—the center of the circle containing dimension lines.



A group of dimensions created by the **Chain Angular Dimension** command, is not an integral object— those are several simple angular dimensions created according to certain rules (the first angle side of each subsequent dimension coincides with the second angle side of the previous one; all dimension lines are located on the same circle). Therefore, to use the same parameters for all dimensions in a chain (for example, a text on a bent leader shelf or arrows of a certain type), after setting up the first dimension enable the **By Default** option.

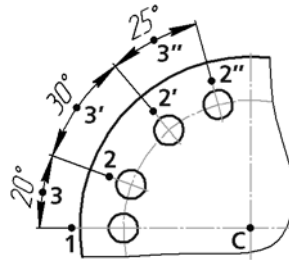


Fig. 3.3.38. Creation of Angular Dimension Chain;
Method of specifying angle sides— **by two line segments**



To specify the next angular dimension chain, click the **Reselect** and select a new angle side.

3.3.4.5. Angular Dimension with Common Dimension Line



To create a group of angular dimension with a common dimension line, call the **Angular Dimension with Common Dimension Line** command.

Specify the first angle side.

After that, specify the second angle side (see Section 3.3.4.1.1 on p. 1005).

The angle vertex and snap points **p1** and **p2** coordinates appear on the Property bar.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

Specify the **p3** point that defines the location of the dimension line. This position will be the same for all dimensions in the group.

Sequentially select the second angle sides for the rest of dimensions in a group.



All segments selected to create an angular dimension with a common dimension line have to pass through one point— the centre of the circle containing dimension lines.



A group of dimensions created by the **Angular Dim with a Common Dim Line** command is not an integral object— it is a chain of angular dimensions whose first angle sides coincide and dimension text is positioned in a special way. Therefore, to use the same parameters for all dimensions in a group (for example, arrows of a certain type), after setting up the first dimension enable the option **By Default**.

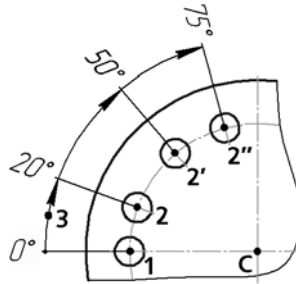


Fig. 3.3.39. Creation of angular dimensions with a common dimension line;
Method of specifying angle sides— **by three points**



To create the next group of chain dimensions, click the **Reselect** button and specify a new angle side.

3.3.4.6. Angular Dimension with Broken Line



To specify an angular dimension with broken line, call the **Angular Dimension with Broken Line** command.

Select the object from which you need to specify the dimension— first side of angle.

After that, specify the dimension axis of symmetry. It is specified in the same way as the second side of the angle (see Section 3.3.4.1.1 on p. 1005). An auxiliary line can be selected as the axis.

The angle vertex and **p1** and **p2** coordinates appear on the Property bar.

Define the **p3** point determining the position of the dimension line and its length.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).

If on-shelf location of the dimension text is selected when setting up the dimension display parameters, specify the starting point of the shelf — **p4**. The leader will start in the middle of the dimension line.

Methods to control the dimension line and location of dimension text are described in Section 3.3.1.5 on p. 987.

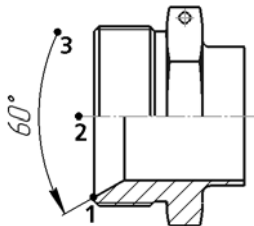


Fig. 3.3.40. Creating an angular dimension with a broken line;
Method of specifying angle sides— **by two line segments**

3.3.4.7. Features of angular dimension editing with the mouse

You can edit an angular dimension changing the location of its handles with the mouse in the document window.

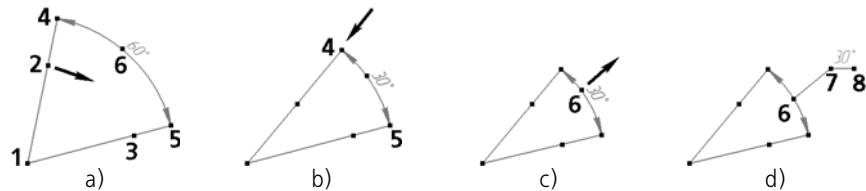


Fig. 3.3.41. Angular dimension editing with the mouse

More information about editing by changing the location of its handles is given in Section 3.4.1.1.3 on p. 1114.

An angular dimension has a several handles. The number and location of handles depends on the method of positioning the text (Fig. 3.3.41).

- 1— a point to change the angle vertex position;
- 2, 3— a points to change the position of angle sides and snap points of extension lines (Fig. a);
- 4, 5— a points to offset dimension lines, (Fig. b);
- 6— the point to move text if positioned manually; the point to move the extension if the text is positioned on the shelf (see Table 3.3.3 on p. 977), (Fig. c);
- 7— a point to move the shelf, (Fig. d);
- 8— a point to change the shelf direction.

3.3.5. Auto Dimensions

Dimensions of frequently used types (simple linear, angular, radial, and some others) are convenient to create with the automatic dimensioning command.



To call the command, click the **Auto Dimensions** button on the **Dimensions** toolbar.

After calling the command, you will have to specify the objects for entering dimension: curves or points. Depending on which objects are specified, the system will automatically determine the type of the dimension created. Therefore, you don't need to call a special command to specify dimensions of each type— it is enough to specify the necessary objects.

The rules for specifying objects while working with the **Smart Dimension**:

- ▼ if, when selecting an object, a defining point of that object got into the cursor's catcher, then it is assumed that the defining **point** rather than the object is selected.
- ▼ if, when selecting an object, none of the defining points of that object got into the cursor's catcher, then it is assumed that the object is selected, which is the **curve** command.
- ▼ if several objects got into the catcher, then selected is considered the object (**point** or **curve**) nearest to the catcher's center.
- ▼ When using the **Nearest Point**, **Midpoint** or **Intersection** snaps, then a selected one is considered to be the **point** rather than the curve to which it belongs.



In order to setup the cursor's "catcher" size, use the **Service — Parameters... — System — Graphic Editor — Cursor** command.

Thus, to create a linear dimension, you need to specify a line segment or two points, between which you need to create the dimension; to create an angular dimension— two nonparallel line segments and so on. For details on specifying objects for dimensions of different types, see the following sections.

If needed, edit the dimension text (see Section 3.3.1.3 on p. 979) and set up the dimension display parameters (see Section 3.3.1.2 on p. 977).



The process of smart dimension editing is run by double-clicking on its image. Editing the parameters set while executing the **Auto Dimension** command isn't different from editing the parameters set after executing any special commands.

3.3.5.1. Linear Smart Dimension

In order to create a linear smart dimension, specify points and/or objects in the document. For more information about object specification when setting dimensions, see Section 3.3.2.1.2 on p. 991.

The linear dimension ghost will be displayed.

By moving the cursor, select the dimension orientation— horizontal, vertical or parallel to an object.

Select the point that will determine the marquee box position.



The linear smart dimension is offered by the system by default after calling the **Auto Dimension** command and specifying a line segment. If you need to create not a simple linear dimension, but rather, for example, a linear dimension from a line segment to a point or an angular dimension, then continue selecting objects. The system will go into the mode of creating a dimension corresponding to the selected objects.

3.3.5.1.1. Methods to Manage Smart Dimension Texts

To control the dimension line and the smart dimension text, use `<Shift>` and `<Ctrl>` keys.

Upon pressing and holding the `<Shift>` key, the dimension line is fixed. By moving the cursor, you can define the position of the dimension text.

Upon pressing and holding the `<Ctrl>` key, the dimension orientation is fixed, and it obtains a leader line and a horizontal flange. Pay attention to the fact that in this case the start point of leader line is the midpoint of the dimension line. By moving the cursor, you can specify the flange start and direction— right or left.

You can use `<Shift>` and `<Ctrl>` keys while creating smart dimension of all the other types. The only exception is radial smart dimensions for which you can only use the `<Ctrl>` key.

3.3.5.2. Linear Smart Dimension from Line to Point

To create a linear smart dimension from a line segment to a point, specify the segment end point or two parallel segments, or a segment and a circle (circle arc).



If one of the selected segments uses the *Centerline* system line style or is a **centerline** system object (see Section 3.3.7.15 on p. 1049), then the dimension type is automatically set to **Linear Dim with Broken Line**. Therefore, if you need to create a linear smart dimension from a line segment to a point, pay attention to the style of the selected segment lines.

The linear dimension ghost will be displayed.

Select the point that will determine the marquee box position.

Additional tools to manage dimension lines and dimension texts are described in the previous section.

3.3.5.3. Linear Smart Dimension with Broken Line

To create a linear smart dimension with a broken line, select two parallel line segments in any order, one of which has the *Centerline* system line style or is a **Centerline** object (see Section 3.3.7.15 on p. 1049).

The linear dimension with broken line ghost will be displayed.

Specify the point determining the position of the dimension line and text.

Additional tools to manage dimension lines and dimension texts are described in Section 3.3.5.1.1.

3.3.5.4. Angular Smart Dimension

To create an angular smart dimension, select two nonparallel straight objects. You can use the following objects:

- ▼ segment (including a polyline link or a polygon side),
- ▼ object of the **centerline** type,
- ▼ auxiliary straight line,
- ▼ axis of the absolute coordinate system or view coordinate system.



If of the specified objects is a line segment of the *Centerline* system style or the **centerline** type of object (see Section 3.3.7.15 on p. 1049), the dimension type will be automatically defined as the **Broken angular**. Therefore, if you need to create a simple angular dimension, pay attention to the style of the selected segment lines.

The angular dimension ghost will be displayed.

By moving the cursor around, select the dimension type: on an acute angle, on an obtuse angle or on an angle greater than 180°.

Specify the point determining the position of the dimension line and text.

Additional tools to manage dimension lines and dimension texts are described in Section 3.3.5.1.1 on p. 1014.

3.3.5.5. Angular Smart Dimension with Broken Line

In order to create a broken angular smart dimension, specify two non-parallel straight objects in any order. You can use the following objects:

- ▼ segment (including a polyline link or a polygon side),
- ▼ object of the **centerline** type,
- ▼ auxiliary straight line,
- ▼ axis of the absolute coordinate system or view coordinate system.

One of the specified objects should be the segment of the *Centerline* system line style or an object of the **centerline** type (see Section 3.3.7.15 on p. 1049).

The angular dimension with a broken line ghost will be displayed.

Specify the point determining the position of the dimension line and text.

Additional tools to manage dimension lines and dimension texts are described in Section 3.3.5.1.1 on p. 1014.

3.3.5.6. Radius Smart Dimension



To create a radius smart dimension, select a circular arc.



The radial dimension ghost will be displayed.

Group of **Type** switches on the **Dimension** tab allows to specify whether to create the radial dimension from the centre or not from the center of the circle.



The **Dimension** group of switches allows to specify whether to create a diameter or a radial dimension. While selecting a diameter dimension, the system goes into the dimension-creating mode (see the next section).

Specify the point determining the position of the dimension line and text.

Additional tools to manage dimension lines and dimension texts are described in Section 3.3.5.1.1 on p. 1014.

3.3.5.7. Diameter Smart Dimension



To create a diameter smart dimension, select the circle.

The diameter dimension ghost will be displayed.



A dimension line can be Continuous or Broken. To select an appropriate variant, use the **Type** group of switches on the **Dimension** tab.



The **Dimension** group of switches allows to specify whether to create a diameter or a radial dimension. While selecting a radial dimension, the system goes into the dimension-creating mode (see previous Section).



When creating a diameter smart dimension, then it is possible to create a linear dimension with the “diameter” symbol (Figure3.3.42, a) or a diameter dimension proper (Figure3.3.42, b). To

specify a linear dimension, you need to move the pointer in the direction of the coordinate axes of the current drawing (for fragment— in the directions of the coordinate axes of the global coordinate system, which is vertically or horizontally); to specify a diameter dimension, you need to move the pointer at an angle to these axes.



Fig. 3.3.42. Circle dimension: a) linear, b) diameter



While specifying an arc diameter dimension, it is impossible to create a linear dimension.

Specify the point determining the position of the dimension line and text.

Additional tools to manage dimension lines and dimension texts are described in Section 3.3.5.1.1 on p. 1014.

3.3.6. Level Dimension



To specify a level dimension, call the **Level Dimension** command.

On the Property bar, there is the **Type** drop-down list, using which you can select a dimension for the desired purpose. The available dimension types are presented in Fig.3.3.43.

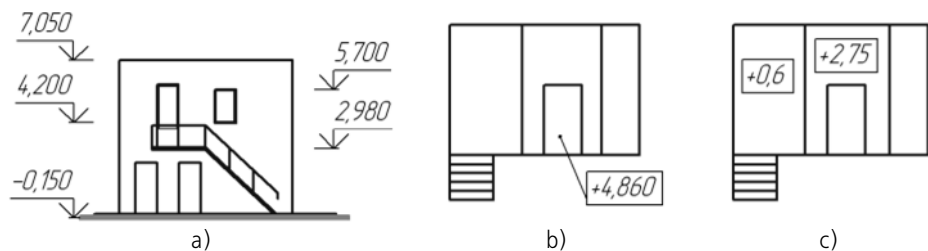


Fig. 3.3.43. Level dimension: a) for a front view or cut, b) for a top view with a leader, c) for a top view immediately in drawing

3.3.6.1. Front View or Cut

Specify the **p0** point which shall be the reference point for level definition (zero level).

Select the snap point **p1** defining the position of the level being dimensioned.

The system will automatically calculate the height value of the specified point with respect to the **t0** point (calculations are run in meters with the precision up to 1/10).

You can edit and automatically determine a value, and also set up the dimension text parameters. To do this, click inside the **Text** field on the Property bar. The dialog to define the text will appear (Fig.3.3.44).

The control elements of this dialog are represented in Table 3.3.10.

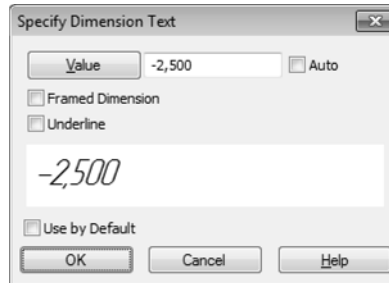


Fig. 3.3.44. Text input dialog for level dimension

Tab. 3.3.10. Text input dialog for height dimension

Element	Description
Value	This field displays the dimension value which was automatically calculated by the system. If necessary, you can enter a different value manually. To select a value from the user menu, click the Value button.
Auto	This option manages the size value specification method. If the option is enabled, the size value is defined automatically. If the value has been input manually, the option shall be automatically disabled. To restore the automatically specified value, enable the Auto option again.
Framed Dimension	The option allows to draw a frame around a dimension value.
Underline	The option that allows underlining a dimension value.
Use by Default	If this option is enabled, all current settings shall be used to create the following dimensions of the current type until the end of the session. If the option is disabled, then the setting affects only the current dimension (the one being created).

While creating a dimension text and setting up its parameters, its current view is presented in the dialog in a special preview field.

Then, select the point **p2** defining the text position.

Select the points **p1** and **p2** for other levels, whose height should be calculated from the same zero level baseline point.

To create a group of level dimensions starting from another zero level, unlock the zero level point (the **p0** fields on the Property bar) and specify its new location.

3.3.6.2. Top View with Leader and Top View Immediately in Drawing

When creating a height dimension on the top view, only the manual text input is possible. Calling and working with the dimension text input dialog are described in the previous section.

To create a height dimension with a leader, specify the point **p1** which defines the level being dimensioned (the leader line will start in it), and then the point defining the position of the dimension text **p2**.

To specify a level dimension immediately in drawing, specify the point that defines the position of the dimension text: **p2**.

3.3.7. Designations for Mechanical Engineering

3.3.7.1. General Information

The commands to place designations in the documents created up to mechanical engineering standards are grouped in the **Tools — Symbols** menu; and the command buttons are located on the **Symbols** bar (Fig. 3.3.45).



Fig. 3.3.45. **Symbols** bar

3.3.7.2. Customizing designations for mechanical engineering in current and new documents

Formatting of a drawing can be much faster if the main parameter set is used as default — creating a new designation, only small corrections will be needed.

KOMPAS-3D uses the default symbol settings that are most commonly used in design documentation.

To change the default designation parameters in the current document, call the command **Service — Parameters... — Current Drawing (fragment)**.

Designations are customized in the following sections and subsections located in the left part of the dialog:

- ▼ **Lines**
 - ▼ **Centerline,**
- ▼ **Lines of Break,**
- ▼ **Leader,**
- ▼ **Intersection Notation,**
- ▼ **Designations for mechanical engineering:**
 - ▼ **Ref. No. Leader,**

- ▼ **Surface finish,**
- ▼ **Shape Deviations and Datum,**
- ▼ **Cut/Section Line**
- ▼ **View arrow,**
- ▼ **Auto-sorting,**
- ▼ **Modification Symbol,**
- ▼ **Overlapping Objects.**

These sections contain control elements that allow setting up the designation parameters in the current document; in some sections, control elements are grouped into subsections.

The setting will be saved in the current document and will not change when opening it on another workplace.



All the setting in the **Leader** section except for those of the arrow filter are also applicable to the stamping and marking symbols. The arrow filter functions for leaders only.



Arrow length and filling as part of the geometric tolerance and positioning are subject to the setup made for dimensions (in the **Dimensions** section of the setup dialog, see Section 9.2.6.10.4 on p. 1773).



Setting up the font height and the symbol (circle diameter, square side, and the dimension designation symbol) size, make sure that the sizes fit.

If you use the same designation parameters in most documents, you can set up the system to create every new document with the desired settings by default.

To do this, call the **Service — Parameters... — New Documents — Graphical document** command. In the left part of the dialog that pops up, there are sections with sets of adjustable parameters for new documents equal to those of the current document.

3.3.7.3. Surface Finish



To create a surface finish designation, call the **Surface Finish** command.

Select with the cursor an object to apply a surface finish symbol (the part silhouette, a dimension extension line, etc.).



By default a surface finish symbol is created, whose machining type is not specified. In the **Type** group on the **Symbol** tab of the Property bar, the option **Treatment Method Not Defined** is active in the Type group. To create a finish symbol for a surface generated with or without material removal, set the appropriate option in this group.



Input the designation text and set up its display parameters.



Specify the point **p** defining the location of the designation.

If the **p** point is specified beyond the object, the location of the symbol will be defined by this point's projection on the object or its extension. In the latter case, the object will be automatically extended as a thin line by the required distance (Fig.3.3.46).

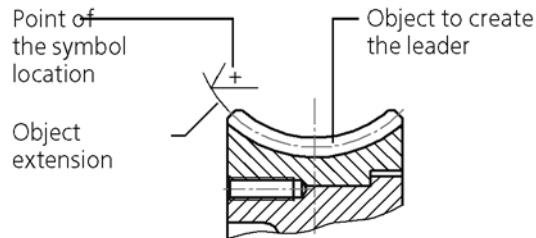


Fig. 3.3.46. Specifying a surface finish value on the extension of an object



Extending NURBS and Bezier curves is not possible.

Use snaps for precise positioning of the cursor (see Section 3.1.1 on p. 865).

If the surface designation's position on the shelf is selected (see Table 3.3.11 on p. 1023), the **p** point defines the leader start point. In this case, to define the symbol position, you need to specify the starting point of the shelf — **p1** (Fig.3.3.47, b).

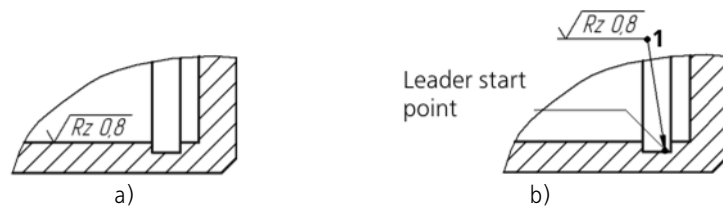


Fig. 3.3.47. Specifying surface finish value: a) without shelf, b) on shelf

The structure of the surface finish symbol is defined by GOST 2.309-73 (Fig.3.3.48).

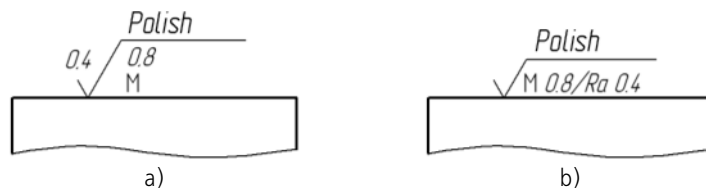


Fig. 3.3.48. The structure of surface finish designations:
a) according to the previous GOST2.309-73 edition,
b) according to modification 3 in GOST2.309-73, 2003

Selecting the structure, used in the current document, is performed in the **Surface Finish** section in the settings dialog of the current document (see Section 3.3.7.2 on p. 1019). By default,

surface finish designations are created in new documents according to the change in GOST2.309–73.



Without leaving the command, you can create several surface finish symbols. Selected symbol type and the generated text will be saved.

3.3.7.3.1. Text of surface finish designation input

If you need to specify only the surface finish height parameter (Ra, Rz, or Rmax) in a surface finish designation, call the context menu in the **Text** field on the **Symbol** (Fig.3.3.49).

Select the desired parameter and its value.

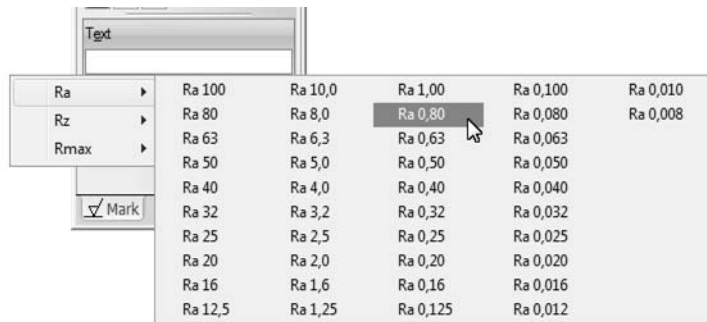


Fig. 3.3.49. Example of surface finish designation selection in the context menu

If there is additional information in the surface finish designation, call the designation text input dialog (Fig. 3.3.50). To do that, left-click in the **Text** field. You can simply start inputting the text; the dialog will automatically appear on the screen.

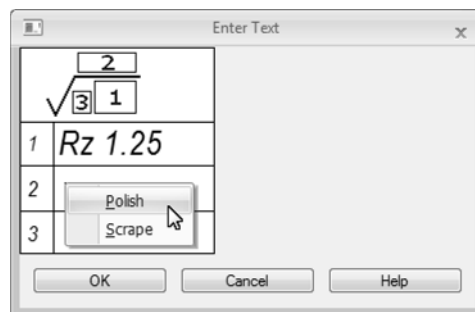


Fig. 3.3.50. Text of surface finish designation input

The surface finish designation and the text structure are shown in the dialog.

Input the desired text.

To accelerate the input of different parts of the text, you can use user menus.

Double-click in any text input field in the dialog to call the corresponding user menu.

In the first field, the user menu contains the surface finish parameters (Ra, Rz, Rmax, Sm, S) and their values, the relative bearing length parameter (p) and the profile section level param-

eter, as well as the values of the base lengths. The second field contains the names of surface processing methods. The third field contains roughness directions. On selecting any name (“Arbitrary”, “Radial”, etc), the corresponding symbol is placed in the surface finish designation.

If needed, change the default text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.




If some text parameters of all surface finish designations in the document are different from the current default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Surface Finish** section in the settings dialog of the current document.

After inputting and formatting the text in the fields, click the **OK** button in the dialog.

3.3.7.3.2. Setting up the Surface Finish Symbol Display

To change the display style of a surface finish designation, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.11.

Tab. 3.3.11. Control elements of surface finish symbol display

Element	Description
 By Contour	The radio button that allows to specify the surface finish designation for a surface that forms a contour.
Leader Shelf	The list that manages the symbol location. Available variants: No Leader Shelf, Left, Right, Top, Bottom.
Leader Parameters	While creating a surface finish designation, the length of the leader is presented in the Length field, and the angle between the leader and the abscissa axis is presented in the Angle field. Inputting these values from the keyboard is possible if automatic creation of objects is off (see Section 1.4.2.10 on p. 93) and only after the designation has been fully created (the shelf starting point has been specified), which is editing of the created leader is actually performed in these fields.
Arrow	The list for leader arrow view selection. The list is available if the surface finish designation is located on a shelf.

3.3.7.3.3. Surface Finish Symbol position control

Surface Finish Symbol may be located directly on the dimension line or on the shelf. Variants of symbol positioning are in the **Shelf** list on the **Parameters** tab of the Properties bar. In the list, select the desired item. Therewith, the initial surface finish symbol position will correspond to the selected variant.

When creating a surface finish symbol you can change symbol position in one of the following ways.

- ▼ Selection of another variant of positioning the surface finish symbol from the list.

- ▼ Moving the cursor rightward, leftward, upward or downward after defining of the start point of the leader line. This method is used in case of selection one of the variants of positioning the surface finish symbol on the shelf. It allows changing shelf direction to a reverse one. For the horizontal shelf arrangement the cursor motion to the right/left is used, for the vertical— up/down.
- ▼ Using the <Ctrl> button and the mouse wheel.

If the variant of the surface finish positioning on the dimension line (without a shelf) is selected, an extension line and a horizontal shelf will appear if the <Ctrl> button is kept pressed. By moving the cursor, you can specify the flange start and direction— right or left.

To rotate the shelf, keep the <Ctrl> button pressed and spin the mouse wheel in both directions. If you spin the mouse wheel backward, the shelf will be rotated clockwise. If you spin the mouse wheel forward, the shelf will be rotated counterclockwise.
- ▼ Enabling/Disabling creation of surface finish symbol on shelf with the **Symbol on Shelf** context menu command.

3.3.7.4. Leader



To create a random leader, call the **Leader** command.

Select the starting point of the first branching of a leader line.

Select the start point **p1** of leader bend.

Then, select the start points of other branchings. Their number is not limited.

Input the text on the leader and specify its display settings.

The ghost of the created Leader is displayed. You can edit or delete any of the branches of the leader without leaving the command dialog (see Section 3.3.7.4.3 on p. 1027).



To fix designation click **Create Object** button.

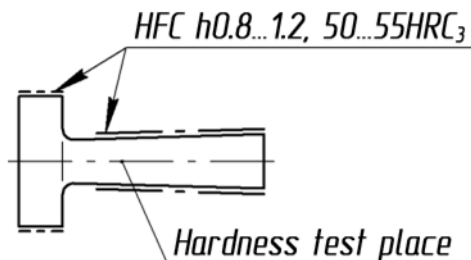


Fig. 3.3.51. Example of leader creation

3.3.7.4.1. Text Input on a Leader

Sometimes, the text consists of an uppercase letter on the leader's shelf. In this case, call the context menu in the **Text** field on the **Symbol** tab of the Property bar and select the desired symbol.

If you need to input a more complicated text, call the text input dialog (Fig. 3.3.52). To do that, left-click in the **Text** field on the **Symbol** tab of the Property bar. You can simply start inputting the text; the dialog will automatically appear on the screen.

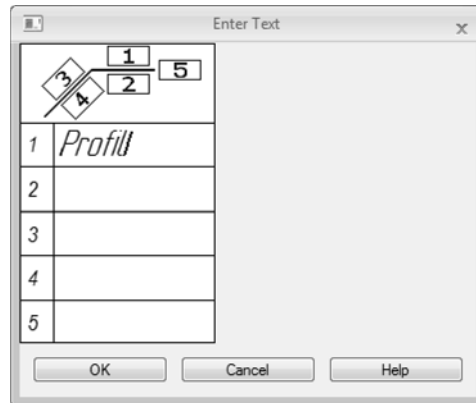


Fig. 3.3.52. The text input dialog on leader.

In the dialog, the structure of leader text is presented.

Input the desired text.

A double click in the first text input field will call a user menu containing uppercase letters.

A double click in the other field allows you to insert a text template (see Section 4.1.2.11.6 on p. 1312).

If needed, change the default text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.







If the text parameters on all leaders in the document are different from the current default parameters, then it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Text Above/Below/After Leader Shelf** and **Inclined Text** subsection in the **Leader** in the settings dialog of the current document.

After inputting and formatting the text in the fields, click the **OK** button in the dialog.

3.3.7.4.2. Setting up Leader Display

To change the leader display style, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.12.

Tab. 3.3.12. Leader Display Control Widgets

Element	Description
Arrow	The list for leader arrow view selection. The list of arrows available for selection, as well as the order of arrows in the list, are defined by the filter settings specified in the Leader — Arrow Filter section in the settings dialog of the current document (see Section 3.3.7.2 on p. 1019).
Type	The list that allows selection of the symbol for a junction designation.
Leader Shelf	<p>The control widget that allows selection of designation shelf location. The Shelf option has different appearance depending on which switch is active in the Branches.</p> <p>If the switch From Shelf Start or From Shelf End is active, Shelf control represents the revealing list, allowing to select a shelf direction. In the list, select the desired shelf display direction: Left, Right, Top, Bottom (Fig.3.3.53).</p> <p>If the Parallel switch is active, then Shelf control represents an option group, allowing to select shelf position — horizontal or vertical (Fig. 3.3.54)*.</p>
Branchings	<p>The group of switches that allows selection of the branching method (Fig.3.3.55). Activate the desired switch:</p> <p>From shelf start,</p>  <p>From shelf end,</p>  <p>Parallel.</p> 
By Contour	<p>The radio button that allows creation of an along contour designation.</p> 
Save Text	<p>If this option is enabled, the text created for the current leader will be used for the rest of the leaders, created during this command call. If the option is disabled, then each next leader line is created without text.</p>
By default	<p>If this option is enabled, all current settings on the Options tab will be used to create the following leader until the end of the working session. If the option is disabled, the setup is only applied to current leader text.</p>

* As long as the type of the **Shelf** control element depends on currently active switch in the **Branches** group, it is recommended to set up leader shelf after the mode of adding branchings has been selected.



Fig. 3.3.53. Shelf direction list

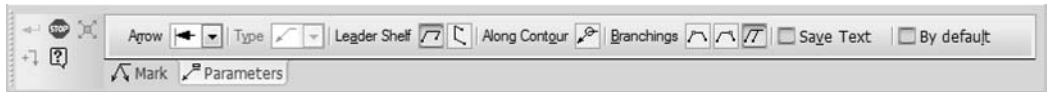


Fig. 3.3.54. Switches to manage shelf locations



Fig. 3.3.55. Leader Length: Leader branches a) from the start and the end of the shelf, b) parallel



To create a leader with branches from the start and the end of the shelf (Fig.3.3.55,a), do the following. Having created the desired number of branches from the start of the shelf, activate the **From Shelf Start** switch and specify the starting points of the branches.



You cannot add branches of the types **From Shelf Start/From Shelf End** to branches created using the **Parallel** switch, and vice versa.

3.3.7.4.3. Changing the Leader Configuration



By default, when creating lines leader, the branch creating mode is on. Pressed **Add Branches** key on Advanced Control panel shows this. In this mode each newly specified point is recognized by the system as the start point of sequential branching. The pointer appears as a cross.



After clicking the **Edit Points** button, the system is switching to the mode of editing object defining points. In this mode, you can modify the symbol's configuration and delete branches; but you cannot add branches. Defining points are displayed as black squares, object— as a ghost. Points in branches' middles are used to create bends (see Pic. 3.3.56).

Move the cursor to a handle of the object. Shape of the cursor will change: it will turn into a four-sided arrow. Press the **<Enter>** key or click the left mouse button, and the point will be selected.

You can edit object points as follows.

- ▼ **Point "dragging" with the mouse.**

▼ **Moving a point from the keyboard.**

For that, the point must be selected by pressing *<Enter>* key.
Move the cursor by the arrow keys.

▼ **Specify the Defining Point Coordinates.**

Specify new coordinates of the selected point on the Properties tab.

▼ **Adding or deleting a defining point.**

▼ To add a point, click the mouse on the required segment of the object. It will be broken down into two parts by a new point in the specified location. You can “drag” it to any position.

▼ To delete a point, select it and click the *Delete* button. This causes the defining point to disappear, and the object will be rebuilt according to the position of the remaining defining points.

If the starting point of the branch is deleted, the whole branch is also deleted.



When defining the position of a point, you can use snaps (see Section 3.1.1 on p. 865) and geometrical calculator (see Section 3.1.2 on p. 871).



The point at the end of the extension line's shelf can only be rotated around the shelf's starting point with a step of 90°.

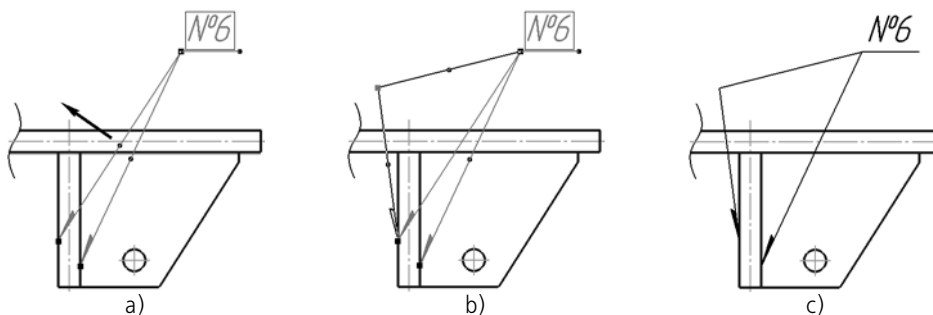


Fig. 3.3.56. Creation of leader bend
a) and b) moving the point in the process of editing, c) bend as a result of editing

You can, at your discretion, switch from one mode of **editing points** and **adding branchings** to another by using the buttons of the Special Control Panel or commands in the context menu. On exiting a mode another one is enabled automatically.

3.3.7.4.4. Changing Symbol Location

You can change the position of a symbol on a leader only after creating the actual leader.

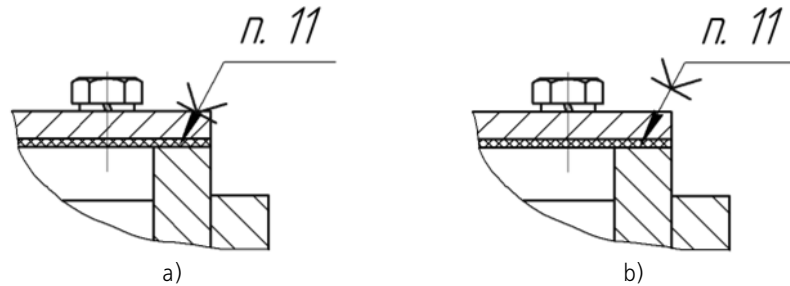


Fig. 3.3.57. Changing symbol position on leader:
a) default symbol position, b) symbol moved up along the leader

To move the symbol, select the created leader by clicking on it. The system will go into the object defining points editing mode. Moving the symbol's handle, place it in the desired location on the leader.

3.3.7.5. Stamping Mark



Serves to create a leader for a stamping symbol the **Stamping Mark** command.

Select the start point of the first branching (the point of placing the first stamp).

Select the point **p1** defining the Stamping Mark position.

Then, select the start points of other branchings.

In the **Text** field on the Property Bar the text is displayed that is suggested by the system as the Stamping Symbol. If needed, you can change the contents and the display style of the text (see Section 3.3.7.5.1).



The **Link** button serves to enable a link into a designation symbol text, for example, a link to a technical specifications item. After clicking the button, the Create Link dialog will appear. (see Fig.4.1.28 on p. 1338 and Table 4.1.24).

Set up the stamping designation display style.

The ghost of the created designation symbol is displayed. You can change its configuration without leaving the command dialog, the same as for leader creation (see Section 3.3.7.4.3 on p. 1027).



To fix designation click **Create Object** button.

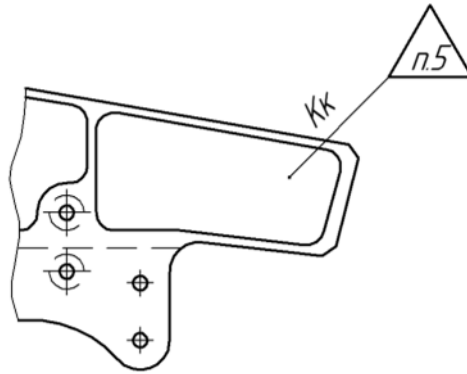


Fig. 3.3.58. Stamp Symbol creation

3.3.7.5.1. Stamping Mark Text Input

Inputting of text is performed in a special dialog (Fig.3.3.59). To call it, left-click in the **Text** field on the **Symbol** tab of the Property bar. You can simply start inputting the text; the dialog will automatically appear on the screen.

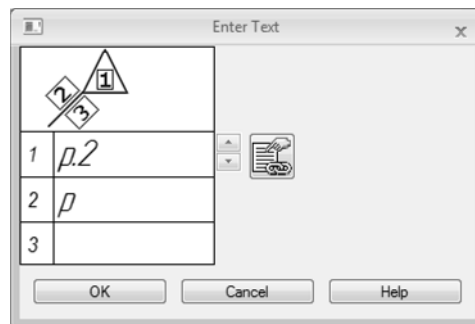


Fig. 3.3.59. Text input dialog for stamping marks

In the dialog, the structure of a stamping designation text is presented.

The buttons with arrows to the right from the first input field allow scrolling the list of system requirements in any direction.

The **Link** button serves to enable a link into a designation symbol text, for example, a link to a technical specifications item. After clicking the button, the Create Link dialog will appear. (see Fig.4.1.28 on p. 1338 and Table 4.1.24).

A double click in the second and the third field calls a special dialog where you can select the contents of the stamping and the display method (in correspondence to the recommended appendix to GOST2.314–68).

If needed, change the default text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.



If the text parameters of all stamping designations in the document are different from the current default parameters, it is recommended to set the desired parameters as default and instead of setting up every single designation text. To do that, use the **Text Above/Below/After Leader Shelf** and **Inclined Text** subsection in the **Leader** in the settings dialog of the current document.

After inputting and formatting the text in the fields, click the **OK** button in the dialog.

3.3.7.5.2. Setting up the Stamping Mark Display

To change the stamping mark display style, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.13.

Tab. 3.3.13. Control elements for stamping mark display

Element	Description
Arrow	The list that allows selection of the stamping designation arrow view.
Direction	The group of radio button that allows selection of symbol drawing direction.
By default	If this option is enabled, all current settings on the Parameters tab will be used to create all following stamping designations until the end of the session. If the option is disabled the setup is only applied to current node point designation.

3.3.7.6. Marking Symbol



You can create a leader for a marking symbol using **Marking Symbol** command.

Select the start point of the first branching (the first point of placing the Marking).

Select the point **p1** defining the marking symbol position.

Then, select the start points of other branchings.

Using the **Arrow** list on the **Options** tab of the Property bar you can select the type of the arrow of the leader line. If this arrow type must be used in the Marking Symbols up to the end of the current working session, enable the option **By default**.

In the **Text** field on the **Symbol** tab, the system-suggested position reference number is displayed. If needed, you can change the contents and the display style of the text. This is made in the same way as while creation of stamping mark (see Section 3.3.7.5.1 on p. 1030).



The **Link** button serves to enable a link into a designation symbol text, for example, a link to a technical specifications item. After clicking the button, the Create Link dialog will appear. (see Fig. 4.1.28 on p. 1338 and Table 4.1.24).

The ghost of the created designation symbol is displayed. You can change its configuration without leaving the command dialog, the same as for leader creation (see Section 3.3.7.4.3 on p. 1027).



To fix designation click **Create Object** button.

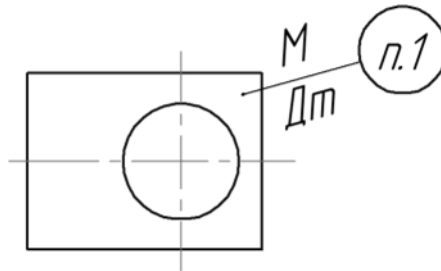


Fig. 3.3.60. Marking symbol creation

3.3.7.7. Ref. No. Leader



Serves to create a leader for a position reference number symbol the **Ref.No. Leader**.

Select the start point of the first branching (the first point indicated by the reference number leader).

Select the start point **p1** of leader bend.



If a reference number designation is creation without a shelf (the option **Shelf** is disabled on the **Options** tab of the Property bar), the **p1** point defines the location of the branching end point.

Then, select the start points of other branchings.

In the **Text** field on the **Symbol** tab of the Properties bar, the system-generated text is suggested — the reference number. If needed, you can change the number and its display style, as well as create additional shelves with reference numbers (see Section 3.3.7.7.1).

Set up the leader display style.

The ghost of the created designation symbol is displayed. You can change its configuration without leaving the command dialog, the same as for leader creation (see Section 3.3.7.4.3 on p. 1027).

To lock the image, click the **Create Object** button.

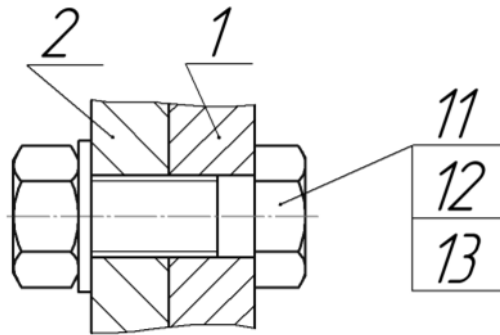


Fig. 3.3.61. Creating position reference number leaders

3.3.7.7.1. Reference Number Text Input

To call the input dialog (Fig.3.3.62), left-click in the **Text** field on the **Symbol** tab of the Property bar. You can simply start inputting the text; the dialog will automatically appear on the screen.

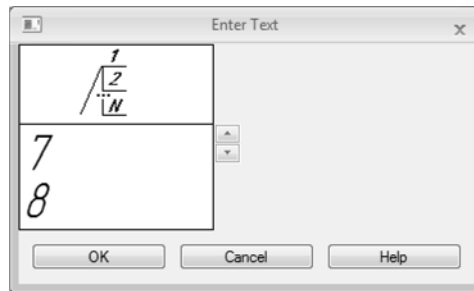


Fig. 3.3.62. Reference number text input dialog

In the dialog, the structure of a reference number text is presented.

The buttons with arrows to the right from the text input field allow you to scroll the reference number list in any direction.

Additional shelves on the leader are created automatically while inputting every new line of the text. To start a new line, press the <Enter> key.

If needed, change the default text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.







If the text parameters of all reference numbers in the document are different from the current default parameter values, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Ref.No. Leader— Ref. No. Text** section in the settings dialog of the current document.

After inputting and formatting the text, click the **OK** button in the dialog.

3.3.7.2. Setting up the Positions Symbol Display

To change the display style of the reference number leader, activate the **Parameters** tab on the Property bar. The control elements located on it are represented in Table 3.3.14.

Tab. 3.3.14. Leader Display Control Widgets

Element	Description
Arrow	The list that allows selection of the arrow view of a reference number leader. The list of arrows available for selections, as well as the sequence of arrows in the list, are defined by the filter settings in the Ref. No. Designation — Arrow Filter section in the settings dialog of the current document (see Section 3.3.7.2 on p. 1019).
 Direction Shelves	This radio group serves to select the direction of displaying the leader line shelf. Changing the shelf direction makes sense only if its display is on (the Shelf option is active). An exception is a position reference number symbol with the form type Simple Text : for it, a change of the shelf direction, when the Shelf option is disabled, means a change in the text position relative to the leader line (left or right).
	
 Text Down/Up	This radio group serves to select the direction of adding position reference numbers. Changing the direction of adding position reference numbers makes sense only if those are arranged vertically (the Horizontal option is disabled).
	
Reference Number Shape	This list serves to select the desired form type for the current position reference number symbol.
Horizontal	This option controls the placement of a symbol that contains several position reference numbers. If the option is disabled, then the numbers are arranged in a column, if enabled— in a row.
Leader Shelf	This option controls the display of the position reference number leader shelf. In the case of reference number positions with the Plain Text form type the shelf appears as a common leader shelf, with the text positioned above it. For other shape types, the shelf is a line segment connecting the text with the leader line (Fig.3.3.63).
By default	If this option is enabled, all current settings on the Options tab will be used to create the following reference number leaders until the end of the session. If the option is disabled, the setup is only applied to current ref.no. leader.

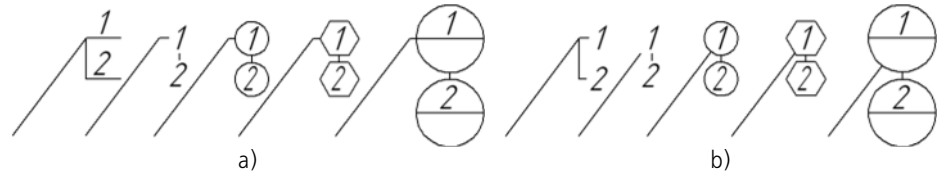


Fig. 3.3.63. Reference numbers of different shapes: a) with leader shelf, b) without leader shelf

3.3.7.7.3. Reference Number Designation Alignment



To place the starting point of several reference number leader shelves in line, horizontal or vertical, select them in the document.



Call either **Align Ref.No.Leaders Horizontally** or **Align Ref.No.Leaders Vertically**—command from the **Tools** menu, or click the respective button on the **Designations** toolbar.

After that, specify the point, whose X or Y-coordinate shall be used to align the starts of the selected leader shelves. You can select a point with the mouse or enter its coordinates in the **Anchor Point** fields on the Property Bar.

3.3.7.8. Modification Symbol



To create a leader for a modification (revision) symbol, call the **Modification Symbol** command.

Select the start point of the first branching (the first point indicated by the leader).

Select the point **p1** defining the modification symbol position.

Then, select the start points of other branchings.

Modifications can be numbered in different ways.

By default, the number suggested by the system is a simple text. And the **Do not use** parameter is selected in the **Numbering** list on the **Sign** tab of the Property bar.

You can set the modification number by placing the created designation into a numbering group. For this purpose, choose the name of the required group from the **Numbering** list. This method makes it possible to number modifications automatically. The number of each change is a link to the designation number of the change in the numbering group. Automatic numbering of objects is described in detail in Section 3.9.1 on p. 1277.

The preset modification number is displayed in the **Text** field of the **Sign** tab. If necessary, you can add a text positioned before or after the number, change the style of characters, number value, if it is a simple text. The process of the designation text modification is described in Section 3.3.7.8.1 on p. 1036.

Set up the modification designation display style.

The ghost of the created designation symbol is displayed. You can change its configuration without leaving the command dialog, the same as for leader creation (see Section 3.3.7.4.3 on p. 1027).



To fix designation click **Create Object** button.

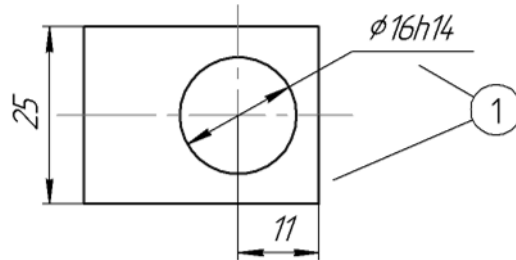


Fig. 3.3.64. Modification (Revision) symbol creation

3.3.7.8.1. Modification Symbol Text Input

To call the input dialog (Fig.3.3.65), left-click in the **Text** field on the **Symbol** tab of the Property bar. You can simply start inputting the text; the dialog will automatically appear on the screen.

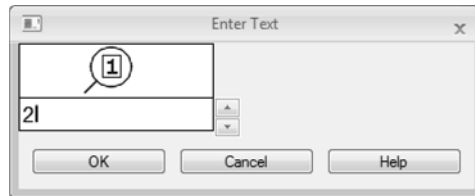


Fig. 3.3.65. The dialog for modification designation text input

The modification's number may be set manually or automatically.

A manually set number is a simple text which can be modified from the keyboard or using the counter located to the right from the text entry field.

An automatically set number is a link to the designation number of the change in the numbering group. It is not available for editing. If necessary, you can delete the link number. The modification's designation will be deleted from the numbering group.

For more information about object numbering, see Section 3.9.1 on p. 1277.

Aside from the modification's designation number, the caption may contain custom text. You can enter a text from the keyboard or insert a text template (see section 4.1.2.11.6 on p. 1312). In order to insert a text template, double-click the text entry field.

If needed, change the default text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.



If the text parameters of all designations of modifications in the document are different from the current default parameter values, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Modification Symbol— Text** section in the settings dialog of the current document.

After inputting and formatting the text in the field, click the **OK** button in the dialog.

3.3.7.8.2. Setting up the Modification Symbol Display

To change the display style of a modification symbol, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.15.

Tab. 3.3.15. Control Widgets for Modification Designation Display Style

Element	Description
Type	The list serves to select the type of the modification (revision) symbol.
Size of Symbol	The field to input or select the symbol size: the length of the square side or the circle diameter *.
Leader type	The list that allows selection of a leader type. If the leader lines of the created Modification Symbol have to be of arbitrary length, select the Full Leader Length option. If you need to create a Modification Symbol with leader lines of the same length, select the Limited Line option and enter the desired value in the Length field.

* If the modification (revision) symbol is set to parentheses, then the **Size of Symbol** field is inaccessible since the height of parentheses is defined by the size of the modification symbol text font.

3.3.7.9. View Arrow



To create a view arrow, call the **View Arrow** command.

Specify the starting point of the arrow — **p1**.



You can easily create a strictly vertical or strictly horizontal arrow in the orthogonal drawing mode. To enable/disable it, use the **Orthogonal Drawing** button on the **Current Status** bar, as well as the <F8> key. To temporarily go into the 'Orthogonal Drawing' mode, press and hold the <Shift> key.

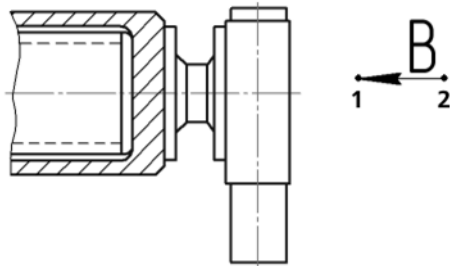


Fig. 3.3.66. Viewing direction arrow creation

The automatically generated text of the view arrow designation will be displayed in the **Text** field on the Property bar. If needed, you can change the contents and the display style of the text (see Section 3.3.7.9.1).

You can choose the arrow style from the **Arrow** list. The list of arrows available for selection as well as the order of arrows in the list is defined by the setting of the arrows filter made for the current document (see Section 9.2.6.10.5 on p. 1774). The arrow drawing setup is described in Section 9.2.6.13.3 on p. 1789.

Select the point **p2** defining the arrow direction.

The arrow will be locked in the document.

Right after creating the direction arrow, the new view creation command is launched automatically (see Section 3.3.14.1 on p. 1110). After the execution of this command, a view will appear in the drawing; its designation will be associatively related to the created arrow.



You can give up creating a new view by clicking the **Interrupt Command** button. However, we do not recommend doing this, because it will take additional time to manually create the view and build the links between its designation symbol and the arrow symbol. Besides that, placing a view in the arrow direction on a separate view helps quickly change the image scale and makes views placement on the drawing more convenient.

While placing a view arrow, a hyperlink may be created automatically to link the designation with the newly created view (see Section 3.3.14.2 on p. 1111).

3.3.7.9.1. Creation of Designation Text

The automatically generated text contains a letter to designate the arrow. The **Auto-Sorting** option indicates, whether the Auto-sorting of object designation symbols is enabled or disabled in the current document (see Section 3.3.12.20 on p. 1067). If the option is active, then you cannot arbitrarily change the letter that designates an arrow. By default, Auto-sorting is enabled.

Upon turning off the **Auto-Sorting** option, you can enter any symbol for the object designation. However, due to disabling this option, automatic sorting does no longer apply to all view arrows in the current document. Activating the **Auto-Sorting** option will turn on the automatic sorting mode again, but the user-created value will be replaced with an automatic one.



You can conveniently pick the designation letter from the context menu of the **Text** field. (Fig. 3.3.67). The set of letters is defines while setting up the automatic sorting engine (see Section 9.2.6.13.14 on p. 1796).

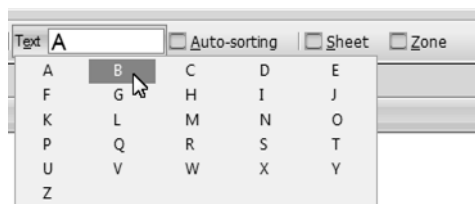


Fig. 3.3.67. Symbol selection menu

If a sheet number, or a designation symbol of the zone, where the view with the respective image will be positioned, needs to be automatically included in the object designation symbol, enable the **Sheet** or **Zone** option.

The sheet number or the zone designation symbol is automatically included into the arrow designation symbol after creating the view and generating an associative relation between the caption of this view and the arrow designation symbol (see Section 3.3.14.1 on p. 1110).



You cannot create sheets and views in fragments. Thus, the options **Sheet** and **Area** are not available.

A layout object designation symbol can be also created in the dialog to enter a special symbol text. To call it, click the left mouse button in the **Text** field. You can simply start inputting the text; the dialog will automatically appear on the screen.

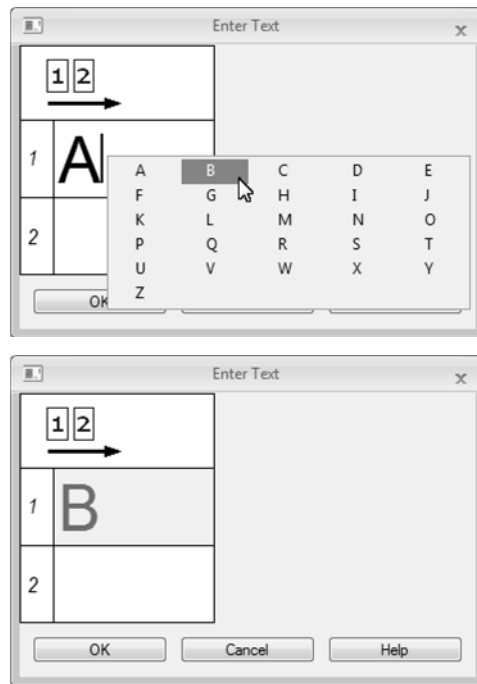


Fig. 3.3.68. Designation's text input dialog

The first input field in the dialog is intended for letters. A double click in this field calls the symbol selection menu (Fig. 3.3.68). In the second field, you can input additional information, for example, area designation.

If needed, change text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.

After inputting and formatting the text in the fields, click the **OK** button in the dialog.



If the text parameters of all view arrows in the document are different from the current default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **View Arrow** section in the settings dialog of the current document.



If the **Auto-Sorting** option is enabled on the Property bar, the first field in the text input dialog won't be available. If the **Sheet** or **Area** option is enabled, the second field won't be available. If the **Auto-Sorting** and **Sheet** (or **Zone**) options are enabled, then the **Text** field itself is inaccessible, and the dialog to enter the text cannot be called.

3.3.7.10. Cut/Section Line



To create a cut/section line, call the **Cut/Section Line** command.

Select the first and second points of the cut line. The specified points are considered to be the start, i.e., positioned closer to the part contour, points of the dashes that depict a cut or section line.

The ghost of a cut/section line designation symbol will be displayed.



If you need to create a complex cut line, click the **Complex Cut** button on the Advanced Control panel. Creation of complex cut lines is described in Section 3.3.7.10.1.

In the **Text** field on the Property bar, the automatically generated text of the cut designation line is displayed. If needed, you can change the contents and the display style of the text. This can be done using the same method as for view arrow creation (see Section 3.3.7.9.1).



To specify at which arrow (first or last) the additional text should be placed, activate the corresponding switch in the **Location Group**. If no additional text is used, then the state of the switches in the group doesn't matter.



You can choose the arrow style from the **Arrow** list. The list of arrows available for selection as well as the order of arrows in the list is defined by the setting of the arrows filter made for the current document (see Section 9.2.6.10.5 on p. 1774). Setup arrows' drawing is described in Section 9.2.6.13.3 on p. 1789.



You can change the configuration of a cut/section line without leaving the command dialog. To do this, click the **Edit Points** button on the Advanced Control panel.

Move the pointer to any handle (these points are displayed as small black squares). Shape of the cursor will change: it will turn into a four-sided arrow.



Change the position of handles by any way or delete unnecessary points, similar to editing a polyline (see Section 3.2.9.1.2 on p. 924)

Move the cursor to specify, on what side of a cut/section line the arrows should be placed. As the cursor crosses the projection of the cut/section line, the ghost will change: the arrows will reposition on the other side of the line.

Click the left mouse button on the side of the line where the arrows should be placed.

The cut/section line will be locked in the document.

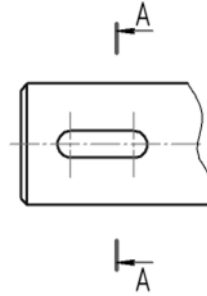


Fig. 3.3.69. Cut line creation

Right after creating the cut section line, the new view creation command is launched automatically (see Section 3.3.14.1 on p. 1110). After executing this command, a view will appear in the drawing. Its designation will be associatively related to the created cut/section line.



You can give up creating a new view by clicking the **Interrupt Command** button. However, we do not recommend doing this, because it will take additional time to manually create the view and build the links between its designation symbol and the cut/section line symbol. Besides that, placing a cut section on a separate view helps quickly change the image scale and makes views placement on the drawing more convenient.

When creating a cut/section line, a hyperlink may be created automatically to link the designation with the newly created view (see Section 3.3.14.2 on p. 1111).

3.3.7.10.1. Creation of Complex Cut Line



To go into the mode of constructing the cut line with a jog or by a polyline, use the **Complex Cut** button on the Advanced Control Panel.

It becomes available after selecting the first point of the cut line.

After clicking that button, each consequent selected point is considered a vertex of the complex cut polyline. For a precise selection, use the snaps or the Geometric Calculator menu in the **Current Point** fields on the **Symbol** tab of the Property Bar. You can also fill in these fields with values by typing on the keyboard.

After selecting a consequent point, the cut line ghost changes (Fig. 3.3.70).

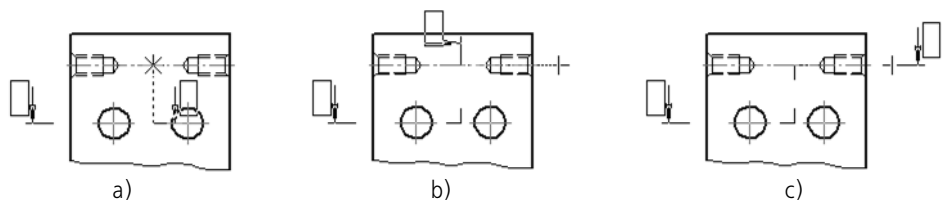


Fig. 3.3.70. Creating a cut with a jog:
a) going into the mode of creating a complex cut line and specifying the third point (the first two points are specified before that), b), c) specifying the fourth point



A cut line, whose segments are perpendicular to each other (for example, when forming a cut with a jog), is easier created in the orthogonal drawing mode. To enable/disable it, use the **Orthogonal Drawing** button on the **Current Status** bar, as well as the <F8> key. To temporarily go into the 'Orthogonal Drawing' mode, press and hold the <Shift> key.

If the orthogonal drawing mode is disabled, then you can create a cut line, whose segments are inclined with respect to each other at an arbitrary angle (for example, when defining a cut by a polyline).

After selecting the last point of the cut line, release the **Complex Cut** button. The system will return on to the simple cut construction mode.

3.3.7.11. Enlarged Detail



To create an enlarged detail designation, call the **Enlarged Detail** command.

The center point of the contour surrounding the enlarged detail.

Specify the contour size: for a circle— diameter or radius; for rectangles— width, height, and rounding radius.

Set up the enlarged detail display style.

In the **Text** field on the Property bar, the automatically generated text of the enlarged detail designation. If needed, you can change the contents and the display style of the text. This can be done using the same method as for view arrow creation (see Section 3.3.7.9.1).

Select the start point **p2** of leader bend.

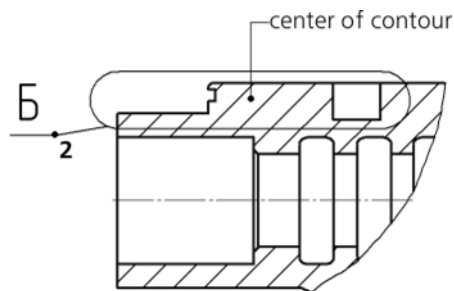


Fig. 3.3.71. Enlarged detail position designation symbol

Right after creating the enlarged detail symbol, the new view creation command is launched automatically (see Section 3.3.14.1 on p. 1110). After the execution of this command, a view will appear in the drawing with its designation associatively related to the created symbol of an enlarged detail.



You can give up creating a new view by clicking the **Interrupt Command** button. However, we do not recommend doing this, because it will take additional time to manually create the view and build the links between its designation symbol and the enlarged detail symbol. Besides that, placing an enlarged detail on a separate view helps quickly change the image scale and makes views placement on the drawing more convenient.

When creating an enlarged detail, a hyperlink may be created automatically to link the designation with the newly created view (see Section 3.3.14.2 on p. 1111).

3.3.7.11.1. Enlarged Detail Designation Display Settings

To change the display style of an enlarged detail designation, activate the **Options** tab on the Property bar. The control elements located on it are represented in table.

Tab. 3.3.16. Control elements to manage the leader element display style

Element	Description
Form	The list that allows selection of an enlarged detail contour shape.
Leader Shelf	This radio group serves to select the direction of displaying the leader line shelf.
By default	If this option is enabled, all current settings on the Parameters tab will be used to create the following enlarged detail designations until the end of the session. If the option is disabled the setup is only applied to current node point designation.

3.3.7.12. Datum



To create a datum surface designation, call the **Datum** command.

Select with the cursor an object that depicts a reference element (the part silhouette, a center-line, etc.).

Specify the **p1** on the base of the triangle that designates the datum.



If the specified point does not belong to the selected object, then the symbol position will be determined by the projection (mapping) of the specified point on the object or its extension. In the latter case, the object will be automatically extended as a thin line by the required distance. To precisely position the cursor, use snaps. For example, if an axis of symmetry is the base, it is convenient to Apply the **Nearest Point** snap, to align the base of the triangle with the end of the dimension line. For details about snaps, see Section 3.1.1 on p. 865.



By default, a datum symbol, perpendicular to the selected reference object, is created. Also, **Perpendicular to Base Object** switch will be enabled **Type** group on the Property bar. If a pitched symbol should be created, set the **Arbitrary Orientation** option.



In the **Text** field on the Property Bar, the letter is displayed that is suggested by the system as the Datum Symbol. If necessary, you can change the symbol, as well as its style (see Section 3.3.7.12.1).

Select the point **p2** defining the text frame position.

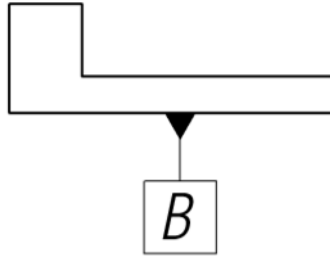


Fig. 3.3.72. An example of applying a datum symbol



The line that connects the datum designation with the frame starts in the middle of the frame's side if the object specified as datum deviates from the vertical or horizontal by 2° or less. If the deviation angle is larger, the line starts at the closest vertex of the frame.

3.3.7.12.1. Datum Designation Text Input

The automatically generated text contains a letter to designate the datum. The **Auto-Sorting** option indicates, whether the Auto-sorting of object designation symbols is enabled or disabled in the current document (see Section 3.3.12.20 on p. 1067). If the option is active, then you cannot arbitrarily change the letter that designates a datum. By default, Auto-sorting is enabled.

Upon turning off the **Auto-Sorting** option, you can enter any symbol for the object designation. However, automatic sorting won't function for all datum designations in the current document in case you disable this option. Activating the **Auto-Sorting** option will turn on the automatic sorting mode again, but the user-created value will be replaced with an automatic one.



You can conveniently pick the designation letter from the context menu of the **Text** field. (Fig. 3.3.73). The set of letters is defines while setting up the automatic sorting engine (see Section 9.2.6.13.14 on p. 1796).

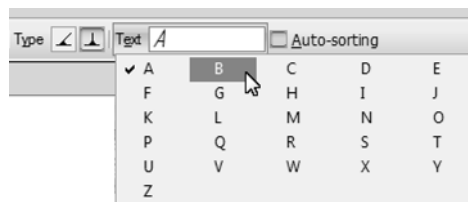


Fig. 3.3.73. Symbol selection menu

A layout object designation symbol can be also created in the dialog to enter a special symbol text.

To call it, click the left mouse button in the **Text** field. You can simply start inputting the text; the dialog will automatically appear on the screen.

A double click in the text input field calls the symbol selection menu (Fig.3.3.74).

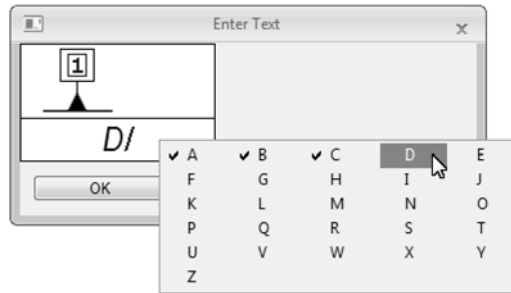


Fig. 3.3.74. The text input dialog for datum designations

If needed, change text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.

After inputting and formatting the text, click the **OK** button in the dialog.



If the text parameters of all datum designations in the document are different from the current default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Shape Deviations and Datum** section in the settings dialog of the current document.



If the **Auto-Sorting** option is enabled on the Property bar, the **Text** field on the Property bar is not available. You cannot call the context menu or the text input dialog.

3.3.7.13. Geometric Tolerance



To create a geometric tolerance designation, call the **Geometric Tolerance** command.

The tolerance frame insertion point. By default, the bottom left corner of the border is placed in the selected point. Also, in the **Anchor Point** list, the **Bottom Left** option is selected. To change the position of the frame relative to the insertion point, expand the specified list and select the desired line.

To make the box appear vertical, turn on **Vertical** option on the Property bar.

Create a tolerance table. The methods and the sequence of its creation are considered below.

Create the desired number of branches with arrows or triangles.

The ghost of the created designation symbol is displayed. You can change its configuration without leaving the command dialog, the same as for leader creation (see Section 3.3.7.4.3 on p. 1027).



To fix designation click **Create Object** button.

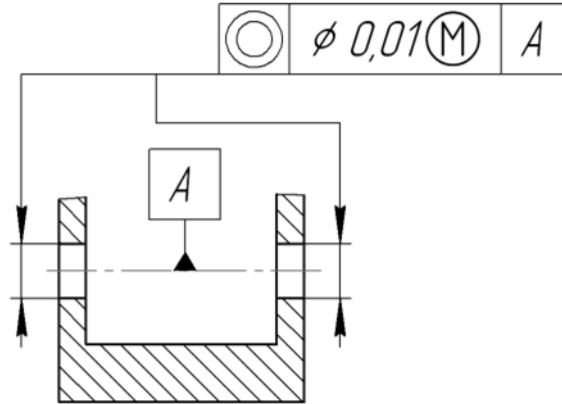


Fig. 3.3.75. Specifying a surface position tolerance

3.3.7.13.1. Creation of Tolerance Tables

There are two methods available for creating a table of the geometric tolerances for surfaces:

- ▼ Semi-automatically
- ▼ manual.



To create a tolerance table in the **Semi-automatic** mode, activate the **Table** switch on the Property bar.

The dialog for text input and designation parameter selection will appear on the screen.

Maximize the **Symbol** sign and select a tolerance designation of the desired type.

Input the numeric value of the tolerance and the designations of the first and the second datum. To accelerate the input, you can use user menus. A double click in any text input field in the dialog calls the corresponding user menu (Fig. 3.3.76).

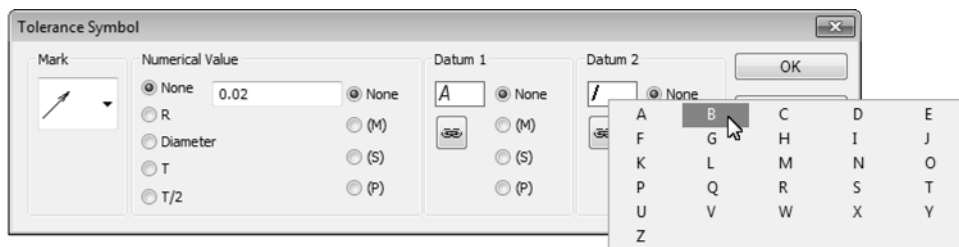


Fig. 3.3.76. User menus for tolerance designation text input



The **Reference** button allows you to insert a reference to a datum designation existing in the document. After clicking this button, a dialog will appear on the screen. In this dialog, you will have to set up the reference parameters (see Fig.4.1.28 on p. 1338 and Table 4.1.24).

Having created the tolerance designation table, click the **OK** button in the **Tolerance Symbol** dialog.

The system will automatically create a table for tolerance designation and fill it in with the selected symbols and values.

To create a tolerance table in the **Manual** mode, click with the mouse on the **Text** field on the Property bar.

The text input dialog for shape and location tolerance designation will appear on the screen.

It contains only one cell. To create a table of a specific structure, use Control elements on the **Table** tab of the Property bar.



The context menu of the cell contains equivalents to almost all commands on the **Table** tab. Using these commands, you can create and fill in a table without using the Properties bar.

Create a table of the desired structure.

Enter text in the table cells. It is convenient to select values and symbols in user menus called by double-clicking in a cell (Fig. 3.3.77).

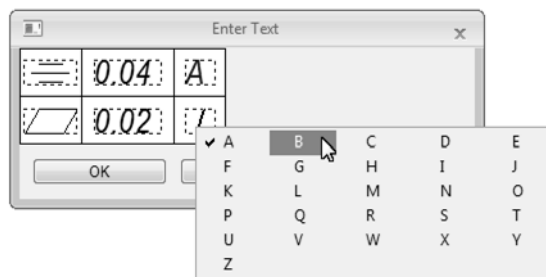


Fig. 3.3.77. Text input in tolerance tables

The user menu in the first cell contains tolerance type names. As you select any of them (co-axiality tolerance, straightness tolerance etc), the correspondent symbol is placed in the designation. The user menu in the second cell contains numeric values of tolerance. The user menus in the third and fourth cells contain capital letters for datum designation.

If needed, change cell text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.



If the text parameters of all shape and position tolerance designation in the document are different from the default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Shape Deviations and Datum** section in the settings dialog of the current document.

When forming a table of the geometric tolerances in the **manual** mode, you can create the link to the existing base symbol in the document using the **Insert— Link** command.

After the creating and formatting the tolerance table, press the **OK** button in the dialog.



A table created in a semi-automatic mode can be edited in a manual mode at any time. For example, the combination of semi-automatic input and manual formatting of text in cells is possible.

3.3.7.13.2. Creation of Branches



To create a branch with an arrow or a triangle, click the corresponding button on the Advanced Control Panel.



On the marquee ghost, eight points will appear, showing the possible places of branching routing.

Click the mouse near the point, where the branching being created should start.



You can easily create branchings, whose segments are perpendicular to each other, in the orthogonal drawing mode. To enable/disable it, use the **Orthogonal Drawing** button on the **Current Status** bar, as well as the <F8> key. To temporarily go into the 'Orthogonal Drawing' mode, press and hold the <Shift> key. If the orthogonal drawing mode is disabled, then you can create branchings, whose segments are inclined with respect to each other at an arbitrary angle.

Select the branching bend points. For precision, use snapping

To finalize a branching creation, click the respective button on the Special Control Panel.

To create the next branching click the appropriate button again.

3.3.7.14. Center Mark



To create a center mark, call the **Center Mark** command.



By default the center mark is created as two intersecting axes. In the **Type** group on the **Center Mark** tab of the Property bar, the **Two Axes** switch will be active. To create a **symbolic center mark** or **one axis**, activate the corresponding switch.



If you need to create a center mark for an axisymmetric object (circle, arc, ellipse, rectangle, regular polygon), specify this object.



If an ellipse, elliptical arc, rectangle or a regular polygon is specified, then the center mark is immediately fixed. The inclination angle in this case is determined automatically.

If a circle or an arc is specified, you need to also specify the inclination angle to lock the center mark.

If you need to create centerlines that don't belong to any symmetrical object, activate the field **Center** or **Angle** on the Property Bar (for this, left-click the mouse on the field name). Then, specify the location of the center point of the mark and its inclination angle.

As a result of executing the command, a special system macro-object is created — the **Center Mark**. Center Mark Axes intersect at the subject curve center always by their dash portions.

By default, the center mark is in no way connected to the object, specified during the course of the operation, and can be edited separately.



Creation of associative (related to the base curve) layout objects is possible while using the parametric mode. For details about this mode and its settings, see Section 7.2.1.7 on p. 1604.

3.3.7.14.1. Setting up Parameters of a Center Mark

To change the center mark display style, activate the **Options** tab of the Property bar. The control elements located on it are represented in Table 3.3.17.

Tab. 3.3.17. Control Widgets for Center Mark Display

Element	Description
Hanging	The field to enter or select the centerline protrusion value from the contour of the part.
Dot*	The field to input or select the Dot Size.
Gap*	The field to input or select the space length between a dotted line and a dash.
Auto Determination	According to GOST, center lines must start and end with dashes. To fulfill this requirement, the dash lengths are proportionately changed when drawing lines. Enable the Auto Determination option if the dash length should fit in the range defined by the standard (5...30 mm).
Dash	The field to input or select the maximum dash length of a centerline. If the value is specified, the standard requirements are not considered. The dash length is calculated to not exceed the specified value. The Dash field is available if the Auto Determination option is disabled.
"Point"	This field serves to enter the length of dashes that make up a cross at the center of the object. This setting works when creating a center mark as a "cross". In the rest of the cases, the length of central dashes is determined by the system automatically.

* According to the GOST 2.303 standard, the combined length of a dash and two spaces must be 3...5mm.



The lengths of elements of centerline and center mark are not depend on view scale, i.e. are set for "paper".

3.3.7.15. Centerline



To create a centerline by specifying two point on it, call **Centerline by Two Points** command. Specify the first and the second point of the centerline (for example, the points of its intersection with the contour of an axially symmetrical part (Fig. 3.3.78,a). In the document, a centerline extending (projecting) beyond the specified points will be created (Fig.3.3.78, b).



Note that after the execution of the **Centerline by Two Points** command, an object named **Centerline** will be created (not a line with the *Centerline* style). Working with objects of this type is not different from working with objects of any other types. For example, using the **Select—By Type**, you can select all centerlines in the document.

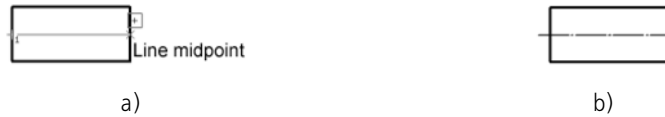


Fig. 3.3.78. Creating centerline by two points:
a) specifying points with snapping, b) result of creation

You can set up the display style of a centerline on the **Options** tab of the Property bar. Control elements on the tab are identical to those on the tab used while creating a center mark (see Table 3.3.17 on p. 1049).

3.3.7.16. Auto Centerline

The **Auto Centerline** command allows you to create a centerline with location and length, defined automatically by the system (depending on the specified objects in the drawing) or by the user.



To call the command, click **Auto Centerline** button on the **Designations** toolbar.

The objects to create an auto centerline. They can be:

- ▼ line segments,
- ▼ points,
- ▼ Symmetrical objects.



To control the creation of a centerline, use the switches in the **Method** group. The **By Items** switch allows creation of a centerline with automatically defined length and location, and the **By Selecting Boundaries** switch is used for a center line of any length.



The options of centerline creation depend on the type of the specified objects, as well as on the selected creation method.

The centerline parameters (jut, dash, and gap lengths) for all options are set up in the same way as in the **Center Mark** command: on the **Options** tab of the Property bar (see Table 3.3.17 on p. 1049).). The only exception is centerline creation for an ellipse (ellipse arc). In this case, you cannot set up the parameters while creating a centerline; they are defined by the centerline settings the current graphic document.

After creating a auto centerline, the system waits until the user specifies the objects for the next auto centerline.



To start editing an auto centerline, double-click on its image.

The process of editing centerlines created using the **Auto Centerline** command is not different from working with objects created using the **Centerline by Two Points** and **Center Mark** commands.

3.3.8. Auto Centerline by Two Points

Creating an auto centerline by two points is very similar to the execution of the **Centerline by Two Points** command.

To create an auto centerline by two points, specify any two points in the drawing. In the document, a centerline extending (projecting) beyond the specified points will be created.

If needed, you can use snapping or auxiliary geometry to specify the points.

The status of the switches in the **Method** group is not relevant while creating an auto centerline.

3.3.9. Creating an Auto Centerline by Items



To create a centerline with automatically defined length and location, activate the **By Items** switch in the **Method** group.

Auto Centerline Creation Relative to a Line Segment

In any sequence, specify the objects for the auto centerline:

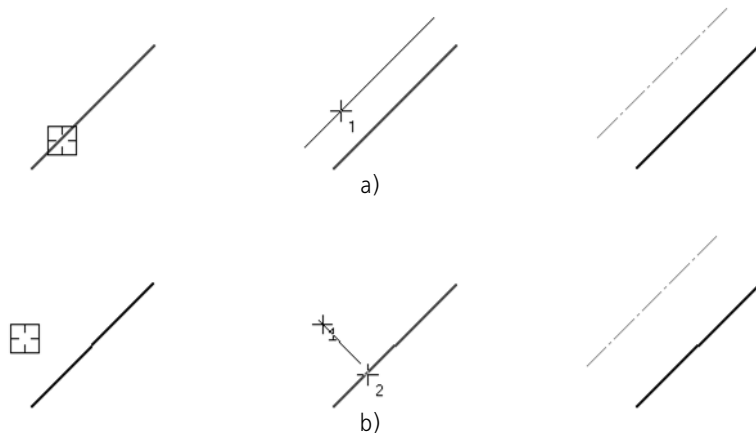


Fig. 3.3.79. Example of auto centerline creation; **By items** method

- ▼ Line, then the point (Fig.3.3.79, a), or
- ▼ The point and then the line (Fig.3.3.79, b).

The centerline will be fixed. Its length is automatically calculated as the sum of the segment length plus the two juts. The location of an auto centerline is also defined automatically. It is placed on the line parallel to the specified segment and containing the specified point; and symmetric to the projection of the middle of the segment on this line.

Creation of an Auto Centerline Relative to Two Segments

If you select two segments as the items to create an auto centerline, the created line will be placed on the bisector of the angle created by them.

Select the first line segment. The centerline ghost, parallel to this line segment will be displayed. Select the second line segment. The centerline will be created automatically. Its position is defined as follows.

1. The position of the start point depends on the relative position of the selected segments:
 - ▼ If the segments intersect, the auto centerline starts at the point of their intersection (Fig.3.3.80,a).
 - ▼ If one segment intersects the imaginary extension of the second segment, the auto centerline begins from the intersection point of these two lines (Fig.3.3.80, b).
 - ▼ If the segments don't intersect, the auto centerline begins in the intersection point of the angle bisector with the imaginary line, connecting the start points of the segments (Fig.3.3.80, c).
2. The end point is the intersection point of the angle bisector with the imaginary line, connecting the end points of the segments selected.

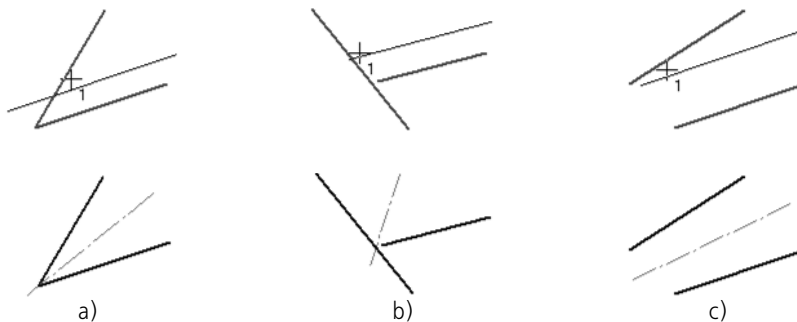


Fig. 3.3.80. Examples of auto centerline creation

If the segments are parallel, the auto centerline will be equidistant from both segments. The start and the end point will be located on the lines that connect the ends of these segments (Fig.3.3.81).



Fig. 3.3.81. Creation of an auto centerline relative to parallel segments

3.3.10. Creation of an Auto Centerline by Selecting Boundaries



To create an auto centerline with an arbitrary length, activate the **By Selecting Boundaries** switch.

Auto Centerline Creation Relative to a Line Segment

In this case, the centerline can be placed either normal or parallel to a line segment.

Select a line segment. Specify the first point, through which the Auto Centerline should go. The system will wait for the selection of the second point on the centerline. The position of this point will define in what way will the centerline be positioned with respect to the line segment—parallel or perpendicular. Move the cursor. Phantoms of the centerline, parallel (Fig.3.3.82, a) or normal (Fig.3.3.82, c) to the selected segment, will appear on the screen.

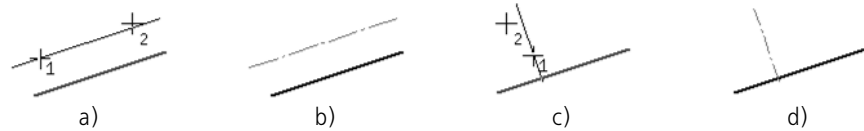


Fig. 3.3.82. Example of auto centerline creation the method named **By selecting boundaries**

Having reached the desired location and phantom length, do a left click. The centerline will be locked (Fig.3.3.82, b, d).



If at the time of selecting the second point the cursor is not positioned on the ghost of the centerline, then the centerline will be bounded by the projection of the cursor position on that line.

To set the second point of the centerline, you can choose a line or a construction line on the drawing (Fig.3.3.83, a). In this case the centerline will end in its point of intersection with the selected line (Fig.3.3.83, b).

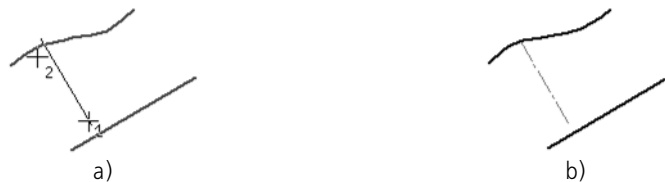


Fig. 3.3.83. Selecting the curve to specify the second point of an auto centerline

Creation of an Auto Centerline Relative to Two Segments

To create an auto centerline placed arbitrarily on a line that contains the bisector of the angle formed by the two segments, do the following.

1. Select the first and second line segments.
2. Specify the start and the end point of the auto centerline (Fig.3.3.84, a).

To specify the limits of the auto centerline, you can select curve or auxiliary straight lines in the drawing. The points of these lines' intersections with the bisector line will be the limits of the auto centerline (Fig.3.3.84, b, c).

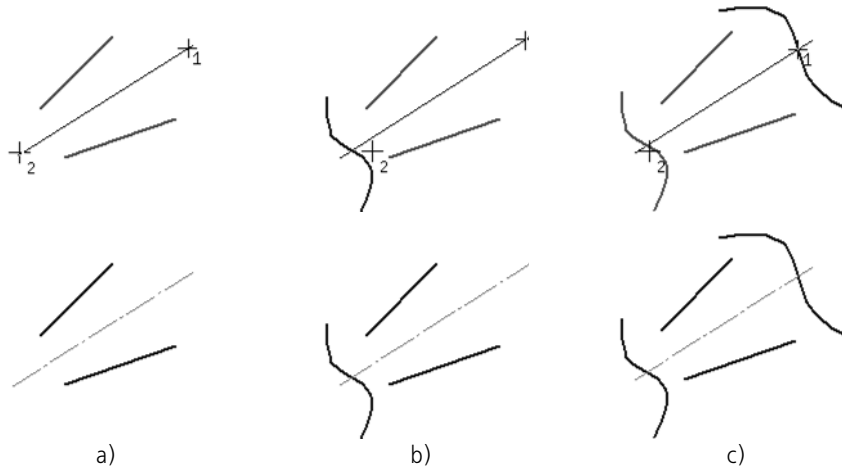


Fig. 3.3.84. Examples of auto centerline creation



If the pointer is not exactly on the bisector's extension while specifying the points, the projections of the pointer's positions onto this line are the limits of the centerline.

3.3.11. Auto Centerline – Center Mark

To create a center mark created by two perpendicular centerlines, select one of the axisymmetric objects:

- ▼ circle,
- ▼ ellipse,
- ▼ arc of a circle or ellipse.

The state of the switches in the **Method** group is not relevant while creating a center mark.



Regular polygons are viewed as sequences of line segments when creating an auto polyline. That's why a polygon cannot be chosen as an object to create a center mark.

If a circle or a circle arc were selected as the object, then a center mark ghost will appear on the screen. As you move the pointer, its rotation angle around the circle/arc center will be changing. To fix the center mark, click the mouse. For precise positioning, use snapping (Fig.3.3.85a, b).

If an ellipse or an ellipse arc is selected as an object, the center mark will be automatically locked (Fig.3.3.85 c, d). The rotation angle of the mark cannot be changed because the centerlines match the axes of the ellipse.

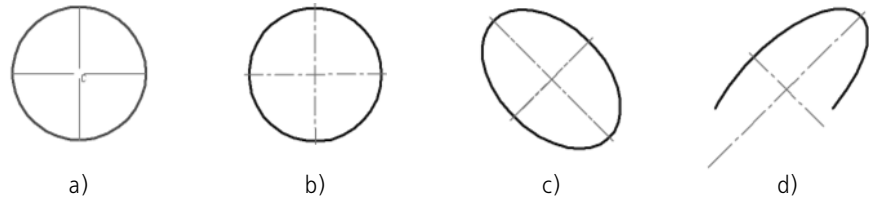


Fig. 3.3.85. Creation of axisymmetric objects' auto centerlines



Working with the **Auto Centerline** command, unlike the **Center Mark** command, you cannot change the designation type.

3.3.12. Example of Auto Centerline Usage

Using the **Auto Centerline** command, you can create centerline images on drawings with minimal numbers of actions.

Example 1: centerline creation with four mouse clicks (Fig.3.3.86). Clicks 1 and 2— selecting segments for the centerline; clicks 3 and 4— specifying the segments that limit the centerline.

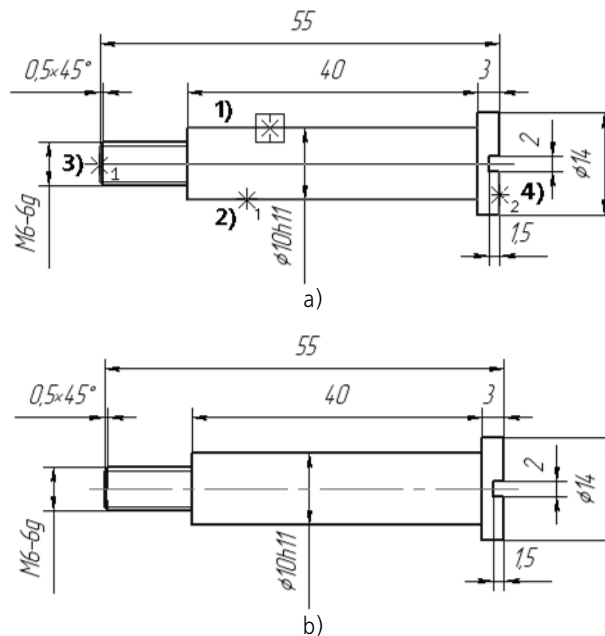


Fig. 3.3.86. Creating an auto centerline using the **By selecting boundaries** method:
a) selecting objects to create an auto centerline, b) auto centerline created

Example 2: creation of a center mark with the **Auto Centerline** command (Fig.3.3.87).

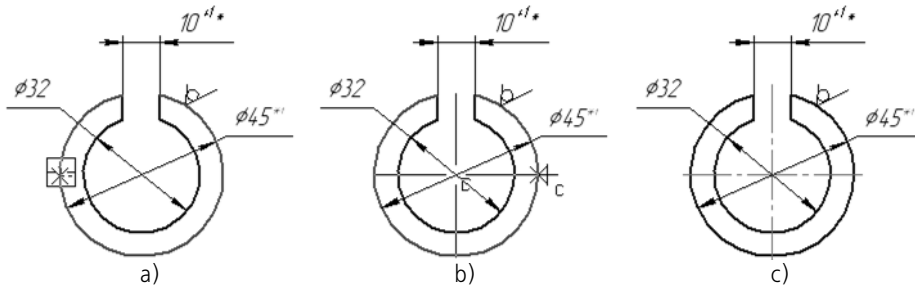


Fig. 3.3.87. Creating a center mark in the drawing:
 a) object selection, b) specifying the center mark location, c) center mark created

3.3.12.17. Wavy Line



To create a wavy break line (Fig.3.3.88), call the **Wavy Line** command.

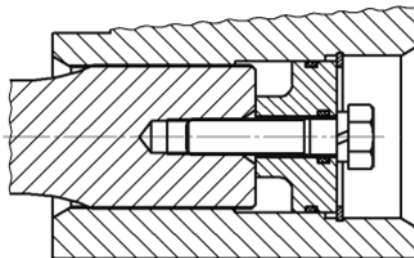


Fig. 3.3.88. Example of wavy line usage

A wavy line is a sinusoid with an even number of half-waves. The start and the end points (**p1** and **p2**) of a wavy line are located on the centerline.

The specific parameters of a wavy line are (Fig.3.3.89):

- ▼ Defining the start and end point,
- ▼ Height,
- ▼ Wave length,
- ▼ Centerline length,
- ▼ The angle between the X axis of the current coordinate system and the centerline,
- ▼ Display direction.

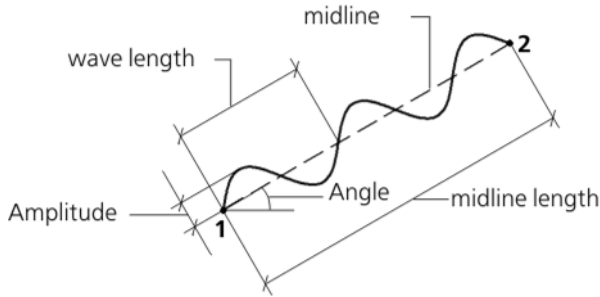
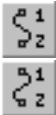


Fig. 3.3.89. Wavy line parameters

To create a wavy line, you can use one of the following methods.

- ▼ Specify the location of the start (**p1**) and the end (**p2**) points of a centerline segment; in this case, the length and inclination angle of a line segment will be determined automatically.
- ▼ Specify the location of the start point of a centerline segment, as well as its length and slope angle. For example, you can specify the **p1** point position with the cursor, enter the length in the field on the Property Bar and define the segment's inclination angle with the cursor. The location of the end point will be defined automatically.

After specifying the **p1** and **p2** points, a wavy line with default parameters will be created in the document.



While creating a wavy line, you can change its display direction with the switch group named **Direction** on the **Wavy Line** tab on the Property bar. In the **Options** tab you can set up the wavy line parameters (see Section 3.3.12.17.1).

To quickly create several wavy lines with the same parameters (for example, to specify an image break), you should do the following.

1. Specify the wavy line parameter values on the **Options** tab.
2. Specify and lock the values in the fields named **Angle** and **Length**. Then, click the **Remember Status** button.
3. Specify sequentially the start points **p1** of the following wavy lines.



You can create a number of designations for one call of the **Wave Line** command.







To finish the creation of wavy lines, click the **Interrupt Command** button on the Advanced Control panel or press the <Esc> key.

3.3.12.17.1. Wavy Line Parameter Settings

To set up the current wavy line parameters, use Control elements on the **Options** tab of the Property bar. These elements are presented in Table 3.3.1.

Tab. 3.3.1. Control elements for wavy line parameters

Element	Description
	Quantity Half-Waves, Wavelength
	
	Height
	Value Amplitude, %
	Value Amplitude, mm
	Style
	By default

* The current or the actual value depending on the creation method, see Section3.3.12.17.2.

The default values of the wavy line parameters for the current document are set in the **Break Lines— Wavy Line** subsection in the settings dialog of the current document. (see Section3.3.7.2 on p. 1019).

3.3.12.17.2. Wavy Line Creation Methods

A wavy line can be created **by the Number of Half-Waves** and **by the Wave Length**. To select a creation method, use the switches named **Number of Half-Waves** and **Wave Length** on the **Options** tab of the Property bar. Activating one of these radio buttons deactivates the other.



- ▼ After activating the **Number of Half-Waves** switch, the **Number of Half-Waves** field becomes available.

This field allows you to input or specify (with the counter) the desired number of half-waves in the line. In the **Wave Length** field, the value of **the current wave length** will be shown in millimeters. The current wave length is calculated as the ratio of the division of the centerline (the distance between points **p1** and **p2** by the number of waves. While changing the pointer's location corresponding to the location of the **p2** point (in other words, while changing the current centerline length), the current wave length is changed, and the number of half-waves remains constant and equal to the specified value.



- ▼ After activating the of the **Wave Length** switch, the **Wave Length** field becomes available. This field allows you to input **the nominal wave length** and reflects **the actual wave length**.

The actual wavelength is the value closest to the nominal value and containing an integer number of half-waves. The actual wavelength is defined as described below.

1. The current length of the centerline is divided by the nominal wave length—the value input primarily in the **Wave Length** field.
2. The ratio is doubled to obtain the number of half-waves.
3. The number of half-waves is rounded to an integer number. The obtained value is presented in the **Number of Half-Waves** field.
4. The current centerline length is divided by the rounded number of half-waves, and the actual wavelength is obtained.

While changing the pointer's location corresponding to the location of the **p2** point (in other words, while changing the current centerline length), both the number of half-waves and the actual wave length are changed. The number of half-waves is always integer, and the actual wavelength varies around the nominal wavelength.



If a value greater than double the distance between the start and end points of wavy line is input into the **Wave Length**, the wave length will be set as double the distance between the ends of the wavy line, and the number of half-waves is set as 1.



Before specifying the **p1** point, the default values are displayed in the fields **Number of Half-waves** and **Wave Length**.

3.3.12.17.3. Wavy Line Editing Features

You can edit a wavy line using the control widgets on the Properties bar or changing the location of its handles with the mouse in the document window. A wavy line has three handles: amplitude point on the first half-wave, and two points at the line ends (Fig. 3.3.90).



Fig. 3.3.90. Wavy line handles

When changing a wavy line amplitude by replacing the amplitude point, number of half-waves and the wavy line length remain unchanged.

When changing a wavy line length by dragging handles 1 and 2, the following parameters may be changed:

- ▼ Ratio of the amplitude to the wave length
- ▼ Amplitude value in millimeters
- ▼ Number of half-waves

Parameter changes depend on the status of the radio buttons on the Properties bar.

The ratio of the amplitude to the wavelength changes if the **Number of Half-Waves** and **Amplitude Value in mm** switches are active (Fig. 3.3.91).



Fig. 3.3.91. Moving the handle **p2** handle with the **Number of half-waves** and **Amplitude value in mm** switches being active

The amplitude value changes if the **Number of Half-Waves** and **Amplitude Value in %** switches are active (Fig.3.3.92).

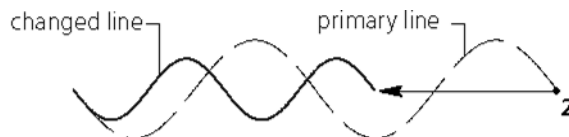


Fig. 3.3.92. Moving the handle **p2** handle with the **Number of half-waves** and **Amplitude value in %** switches being active

The number of half-waves changes if the **Wave Length** switch is active. If the **Amplitude Value in %** switch is active, the ratio of the amplitude to the actual wave length doesn't change (Fig.3.3.93); if the **Amplitude Value in mm** switch is active, the absolute value of the amplitude is not changed.

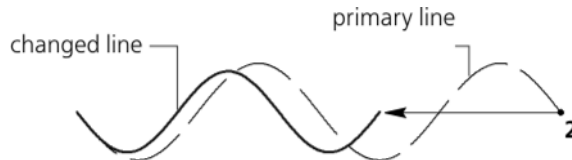


Fig. 3.3.93. Moving the handle **p2** handle with the **Wave Length** and **Amplitude value in %** switches being active.

If the **Wave Length** switch is active and handles 1 and 2 approach each other to a distance which is less than half the nominal wavelength, only one half-wave of the actual length is created (Fig.3.3.94). At that, the value of the nominal wavelength does not change.

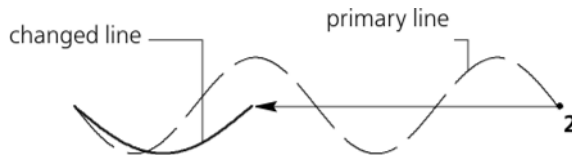


Fig. 3.3.94. Placing the handles at a distance less than half the nominal wavelength

While executing the **Scaling** command, a wavy line will be changing in the same way as during the editing process if the corresponding switches are active (see Fig. 3.3.91– 3.3.94).

To edit a wavy line with the commands used for geometric objects (**Trim Line**, **Split curve** etc.), you need to convert the line to a spline curve - NURBS (see Section 3.4.4.4 on p. 1132).

3.3.12.18. Line with Bends



To create a break line with bends (Fig. 3.3.95), call the **Line with Bends** command.

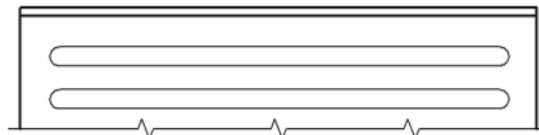


Fig. 3.3.95. Example of using a line with bends



A line with bends is a segment with evenly placed bends.

A **Bend** is a designation fragment with two (type 1) or three (type 2) vertexes.

An **Amplitude** is the distance between the line and the most distant bend point.

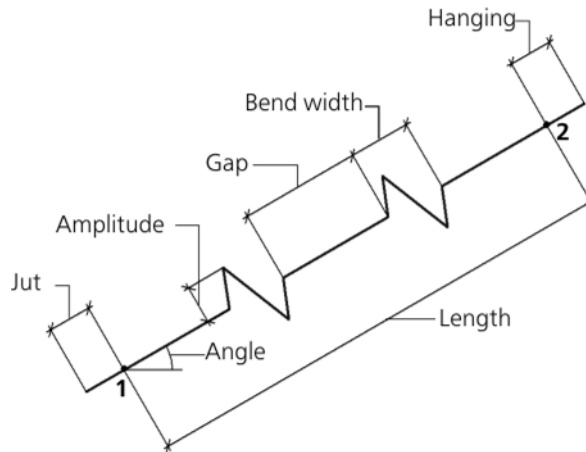


Fig. 3.3.96. Parameters of a line with bends

The specific parameters of a line with bends are (Fig.3.3.96):

- ▼ The coordinates of the start (**p1**) and end (**p2**) points
- ▼ Bend amplitude
- ▼ Polyline length
- ▼ Jut
- ▼ Gap between bends
- ▼ Angle between the X axis of the current coordinate system and the line with bends
- ▼ Type of bends
- ▼ Number of bends
- ▼ Bend displacement from the symmetrical location on the line.

To create a line with bends, you can use the following methods.

- ▼ Specify the start **p1** and end **p2** points; the length and the slope angle of the line will be automatically determined.
- ▼ Specify the location of the start point of the line with bends, its length and slope angle. For example, you can specify the **p1** point position with the cursor, enter the length in the field on the Property Bar and define the segment's inclination angle with the cursor. The end point's location will be automatically determined.

A line with bends will be created in the document. It will extend beyond the limiting points (**p1** and **p2**) by the jut value.

If the specified number of bends with the current parameters doesn't fit in between the line's end points, the maximum possible number of bends is created.

While creating a line with bends, you can set up its parameters on the **Options** tab of the Property bar (see Section 3.3.12.18.1).

To create a few identical lines with bends, for example, to indicate an image break, do the following.

1. Without specifying the **p1** and **p2** points, set up the parameters on the **Options** tab.
2. Specify and lock the values in the **Angle** and **Length** fields.
3. Click the **Remember Status** button.
4. Specify consistently index points (**p1**) new break lines.



You can create a number of designations for one call of the **Bend Line** command.



To stop execution of the command, click the **Interrupt Command** button on the Advanced Control panel or press the <Esc> key.





You can use snapping to a line with bends while creating other objects. You need to keep in mind that the **Midpoint** snap type allows you to snap to the middle of an interval, and the **Center** snap type allows you to snap to the center of the line.

3.3.12.18.1. Setting up the Parameters of a Line with Bends

To set up the parameters of a line with bends, use Control elements on the **Options** tab of the Property bar. These elements are presented in Table 3.3.2.

Tab. 3.3.2. Control elements to manage lines with bends

Element	Description
Hanging	The field that allows you to input or set (with the counter) the value of the line jut beyond the start and end points.
Quantity bends	The fields that allows you to input or set (with the counter) the number of the bends. The minimum number of bends is 1.
Height	The field that allows you to input or set (with the counter) the bend amplitude. As the amplitude changes, the bend width adjusts proportionally.
Offset	The field that allows you to input or set (with the counter) the bend displacement (see Section 3.3.12.18.2).
 Bend Type	The group of radio buttons that allows selection of a mark display style: with two or three vertices. As you change the type, the gap size between bends doesn't change.
 Style	In this fields the current line display style is shown. Expand the list and select necessary line to change the style. The list of styles available for selection, as well as their order in the list, are defined by the line filter settings in the Break Lines— Line Filter section in the settings dialog of the current document (see Section 3.3.7.2 on p. 1019).

Tab. 3.3.2. Control elements to manage lines with bends

Element	Description
By default	If this option is enabled, all current settings on the Options tab will be used to create the following lined with bends until the end of session. If the option is disabled, the setup is only applied to current leader text. Next time you call the command, the default settings will be used.

The default parameters of lines with bends in the current document are set in the **Break Lines— Line with Bends** subsection in the settings dialog of the current document (see Section 3.3.7.2 on p. 1019).

3.3.12.18.2. Bend Displacement

Bend displacement (c) is equal to the percentage ratio of bend displacement value (b) to the gap length between bends (a):

$$c = (b/a) \cdot 100$$

- ▼ If the distance b is zero (the displacement c is then zero, too), the bends are placed at an equal distance from each other and from the line ends (Fig. 3.3.97, a).
- ▼ If the b value is not zero (the displacement c is not equal to zero), the bends are displaced from the symmetrical position. The displacement direction depends on its sign.
 - ▼ $c > 0$: the bends are moved toward the end point **p2** (Fig. 3.3.97, b).
 - ▼ $c < 0$: the bends are moved toward the start point **p1**.
- ▼ If the b distance is equal to the gap between bends (the displacement is equal to 100% or -100%), the bend limit matches the end (Fig. 3.3.97, c) or the start (Fig. 3.3.97, d) point.

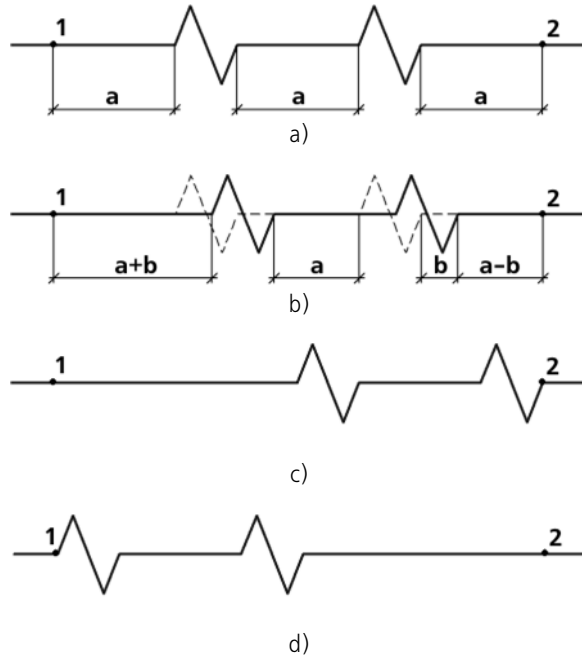


Fig. 3.3.97. Bend Offset



The value of line just beyond the start and end points (**p1** and **p2**) doesn't change while displacing bends.

3.3.12.19. Conditional intersection

Conditional crossing– a symbol of the object extension crossing with another object or its extension. Object extensions are created along the extension lines. A point is created at the intersection of the extension line with an object or another extension line (Fig.3.3.98).

You can create an intersection notation of the following objects (including if they are macro elements and insertions): line segments, circular and elliptical arcs and also equidistant lines of segments and circular arcs.

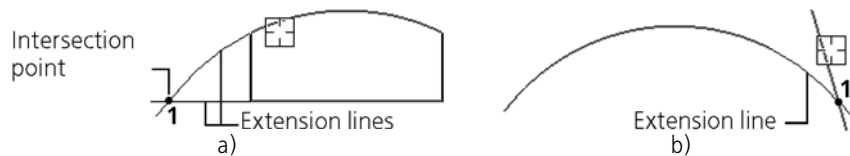


Fig. 3.3.98. Intersection Notation: a) for two objects extensions; b) for an object and other object extension



To create an intersection notation, call the **Intersection Notation** command.

Using the cursor, specify the first and the second object closer to the starting point to be used as the beginning of its extension. Fixing the cursor is done by clicking the left mouse button or pressing the <Enter> key.



The intersection points of the objects themselves, not their extensions, cannot be constructed. To run the command it is required that at least one element from the selected pair must have an extension.

After specifying the first object, it is highlighted. When the second object falls into the cursor trap, the ghost of conditional intersection will be displayed. This occurs if at least one construction method exists for the given pair of objects.

If there are several points of intersection, move the cursor without fixing it closer to another ending point of the object to display another ghost variant. After the ghost appearance in the required point, fix the cursor again.

The size of the cursor catcher can be changed in the Cursor Properties setup dialog (see Section 9.1.7.1 on p. 1703).



Conditional crossing of trimmed objects can be built when creating a chamfer or a rounding (see Section 3.2.15 on p. 970). You can also use the **Conditional intersection** command when creating object by calling it from the geometrical calculator menu (see Section 3.1.2 on p. 871).

By default the created cosmetic intersection is in no way connected to the objects, specified during the course of the command, and can be edited separately.



Creation of associative (related to the base curves) layout objects is possible while using the parametric mode. For details about this mode and its settings, see Section 7.2.1.7 on p. 1604.

3.3.12.19.1. Conditional intersection parameters set up

To set up the parameters of an intersection notation, use Control elements of the Properties bar.



To enable or disable the display of the object extension lines, use **Extension Line 1** and **Extension Line 2** switches.

The extension line may start at some distance from the object point.

You can fix both the length of extension lines and the gap between the object and the start of the extension line. To do this, enter the desired value in the **Gap or length (mm)** field or specify it by counter. Only integral values of length (gap) are allowed. In case of a zero value (or a value greater than the distance between the object and the **t1** point), extension lines are built from ending points of the objects to the intersection point (Fig.3.3.98).



Fig. 3.3.99. Drawing extension lines when setting a non-zero value:
a) gap; b) length



If you need to set the gap between the end of the object point and the start of the extension line, activate the **Gap** switch (Fig. 3.3.99, a).



If the specified value should define the length of the extension line, activate the **Length** switch (Fig.3.3.99, b).

Enable the **Point** option to create a point at the intersection of the extension lines. Select its style of drawing from the drop-down list. If the option is disabled, the point is not created.

Enabled **By default** options means that all current settings will be used when creating new intersection notations through the end of working session. When the option is disabled, the settings affect only the current intersection (the one being created).

Default values of the conditional intersection parameters for the current document, including margins of the extension lines with respect to the intersection point, are set in the **Conditional intersection** section of the settings dialog of the current document (see Section 9.2.6.12 on p. 1785).

In the conditional intersection editing mode, coordinates of the extension lines' intersection point **t1** are displayed on the property bar. Change of coordinates is accessible if the conditional intersection does not have an associative link with objects.

3.3.12.20. Auto-Sorting of Letter Designations

Auto-sorting of letter designations serves to automatically order the letters used in the following designations:

- ▼ a view arrow,
- ▼ Enlarged Details
- ▼ cut lines (for mechanical engineering),
- ▼ Datums.

When creating layout objects, for which the Auto-sorting is enabled, an arbitrary letter cannot be selected. The appropriate letter is picked by the system automatically according to the settings. Once all letters in the list are used up, designations will be assigned to the same letters and in the same order, but with an additional numerical subscript: A_1, B_1, \dots ; then A_2, B_2, \dots and so on up to A_g, B_g .



If, while setting a color for the text elements (see Section 9.1.3.5 on p. 1681), the option **Letter symbols being sorted** is enabled, the letter symbol font color shall correspond to the selected color. If this option is disabled, the font color is black.

Upon deleting one or several layout objects, the remaining objects will be assigned new designations, accounting for the “freed” letters.

If the auto-sorting is disabled for objects of some type, then, when creating such objects you can enter an arbitrary symbol, even if that was already used in another designation. The symbols already in use are ticked in the symbol selection menu (see Fig.3.3.67 and 3.3.68).

Consider an example of setting designations in the auto-sorting mode. For this, do the following.

1. Create a fragment.
2. Make sure that the auto-sorting is enabled in the current document and is using default settings. To do that, call the automatic sorting settings dialog for the current fragment (see below) and make sure that:
 - ▼ All options are enabled
 - ▼ Letters list: **ABCDEFGHIJKLMNOPQRSTUVWXYZ**,
 - ▼ The objects list is:
 - ▼ **View arrow**,
 - ▼ **Enlarged detail**,
 - ▼ **Cut/section line**,
 - ▼ **Datum symbol**.

Otherwise, change settings and click **OK** to close the dialog.

3. Call the **Enlarged Detail** command. The **Text** field on the Property bar will be inaccessible while showing the letter *A* — the first in the letters list; the **Auto-Sorting** option is enabled. These are the signs of the Auto-Sorting mode being enabled (the same-name option is active in the properties dialog).
4. Do not disable the **Auto-Sorting** option. Create an Enlarged Detail.
5. Create another Enlarged Detail. The latter will be automatically denoted with the letter *B*.
6. Select a view arrow. You will see that the letter *A* “switches” to the view arrow, while the Enlarged Detail designations will receive the letters *B* and *C* instead of *A* and *B* respectively. This happened because the **View Arrow** object is positioned higher in the list of objects than the **Enlarged Detail**.
7. Delete the enlarged detail denoted by letter *B*. You will see that the remaining enlarged detail has received the designation *B* instead of *C*, since the letter *B* is “freed” as a result of the deletion.

3.3.13. Designations for Civil Engineering

3.3.13.1. General Information

KOMPAS-3D allows creation of the following designations used in industrial construction design:

- ▼ Marks/reference designations (without a leader, with a leader, on a line)

- ▼ cut line symbols,
- ▼ Sectional node point designation
- ▼ Node point numbers
- ▼ Leader text for multi-layer structures
- ▼ Braces for fragment designations
- ▼ Coordinate axes (straight, angular, circular).

The commands to place designations in the documents created up to civil engineering standards are grouped in the **View — Toolbars — Designations for Civil Engineering** menu; and the command buttons are located on the **Designations for Civil Engineering** bar (Fig. 3.3.100).

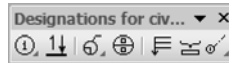


Fig. 3.3.100. **Designations for Civil Engineering** bar

3.3.13.2. Setting of symbols for civil engineering in current and new documents

Formatting of a drawing can be much faster if the main parameter set is used as default — creating a new designation, only small corrections will be needed.

KOMPAS-3D uses the default symbol settings that are most commonly used in design documentation.

To change the default dimension parameters of a construction designation in the current document, call the command **Service — Parameters... — Current Drawing/Fragment**.

Open the **Designations for Civil Engineering** section in the left part of the dialog that pops up. It contains subsections with names matching the names of the designations:

- ▼ **Mark/reference designation,**
- ▼ **Cut Line,**
- ▼ **Node and sectional node designation,**
- ▼ **Node number,**
- ▼ **Leader text,**
- ▼ **Brace,**
- ▼ **Coordination Axes.**

Each subsection includes groups of settings that allow setting designation parameters in the current document.

The setting will be saved in the current document and will not change when opening it on another workplace.



If changes in designation settings in the current document are not desirable, you can set any parameter combination that will be used as default while creating designations until the end of the session.

To do that, enable the Default option (on the **Options** tab of the Property bar) while creating designations of all type (except for **marks/reference designations on line and** node numbers).

If you use the same designation parameters in most documents, you can set up the system to create every new document with the desired settings by default.

To do this, call the **Service – Parameters... – New Documents – Graphical document** command. Open the **Designations for Civil Engineering** section in the left part of the appearing dialog. The list of subsections and the set of adjustable parameters for new documents are the same as for the current document.

3.3.13.3. Mark/Reference Designation without Leader



To create a mark/reference designation without a leader, call **Mark/Reference Number Designation without Leader**.

After calling the command, a designation ghost will appear. The anchor (**p**) is allocated at center of a dimensional rectangle.

Enter the coordinates of the mark/reference designation on the Property bar or specify the point defining its location in the document window.

To construct the turned denotation, enter the necessary value of a corner of turn of the text in the field **Angle**.

Input the designation text (see Section 3.3.13.3.1 on p. 1071).



To do automatic assignment of element numbers while inputting text, activate the **Auto Numbering** switch. For details about auto numbering see Section 3.3.13.3.2 on p. 1073.



Contextual menu of the **Text** field contains all marks saved in the document. To create mark with already existing text designation you can select it from this menu (Fig.3.3.101). It will be assigned sequential number in the mode of auto numbering.

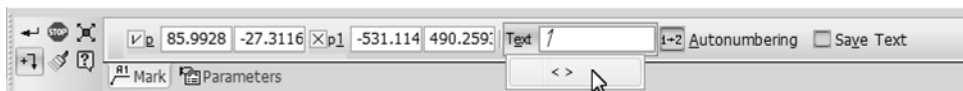


Fig. 3.3.101. The context menu of the **Text** field of a mark/reference designation.

If text entered for the current mark/reference designation (the one being created) shall be used for the rest of the same designations created within this command call, enable the **Save Text** option. If the option is disabled, then each next designation is created without text.

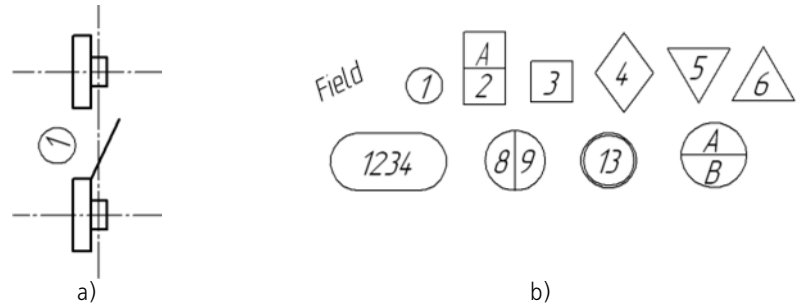


Fig. 3.3.102. Mark/positional designation without a leader
a) example of usage, b) designation form type

Set up the parameters of marks/reference designations on the **Options** tab of the Property bar (see Section 3.3.13.3.3 on p. 1074).

Text can be link source for another text object in this document. For details about links, see Section 4.1.4.3 on p. 1337.



The **Copy Properties** button on the Advanced Control panel allows you to copy text into the current designation from an existing mark/reference designation (of the same or other type). When copying properties, the designations' form type is also transferred.



To fix designation, click the **Create Object** button on the Advanced Control Panel.

You can create several designations during one call of the **Mark/Reference Designation without Leader** command. Specify new anchor point (**p**) after you have clicked **Create Object** button.



The background under the denotation is always cleared irrespective of the customizations made in dialog of parameters of mapping of overlapped objects.

3.3.13.3.1. Mark/Reference Designation Text Input

To input a text while creating and editing the following types of objects:

- ▼ Mark/reference number designation without any leader,
- ▼ Mark/reference designation with a leader,
- ▼ mark/reference designation on line

Use the dialog shown in Fig.3.3.103. The dialog is called by clicking in the **Text** field on the **Symbol** tab of the Property bar by using the **Text...** command. in the context menu of the drawing field.

The control elements of this dialog are represented in Table 3.3.3.



Fig. 3.3.103. Dialog to enter text of the mark/reference designation

Tab. 3.3.3. Dialog to enter text of the mark/reference designation

Element	Description
Text Before	After inputting the text preceding a mark/reference designation.
Mark	The fields for mark designation input. The text of a mark designation is input into the right field, and the number is input into the right field. If there are marks/reference designations in the drawing, the mark can be selected from the list expanding at click on the button with the arrow at the left field. After the mark has been selected (including with no symbols) the list of numbers for current mark becomes available in the right field. The mark number can be selected from a list or input manually. If Auto Numbering of marks is enabled (see Section 3.3.13.3.2 on p. 1073), the following number is suggested for the selected mark.
Text After	The field to input the text following a mark designation.
Text Under	The field to input the text under a designation shelf.
Comment	The field to input auxiliary information, for example, about the element the current mark/reference number is being created for. The comment will not be displayed to the drawing.

While you are creating a designation, its current appearance (except for the text below the shelf) is shown in a special field in the lower part of the dialog window.

Between a mark and an element number, there can be a divider: one, two, or three symbols that are also shown in this field (in Fig. 3.3.103, the divider is a hyphen). Spacer is pasted automatically. If one of designation element is absent (mark name or its number) then the separator is not displayed. The separator is not part of the text and cannot be edited in the text input dialog.

Character input is performed in the **Mark/Reference Designation— Common Settings** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

Dialog menu contains commands for editing, inserting and formatting text. These commands are used in the same way as when working in the text editor (see Section 4.1.2 on p. 1293). Some of them can be called from the context menu of the dialog's input field.



For mark designations, it is recommended to use characters that can be input immediately from the keyboard— digits, Latin and Cyrillic letters etc. If a mark designation contains characters input using the **Insert— Symbol...** and **Insert— Special Character** commands, the designation will be created, but only the part of the designation placed after the first symbol of this type will be shown in the drop-down list in the left field.

After text of the mark/reference designation has been entered click the **OK** button.



If the text parameters (font, height, color etc.) of marks/reference designations (all or of a certain type) in the document are different from the default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Mark/Reference Designation— With Leader/ On Line/ Without Leader— Text** items in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.3.2. Autonumbering of Marks/Reference Designations

1→2

The **Auto Numbering** switch allows automatic assignment of a number to a created mark. The switch is accessible at mark/reference designation creation and is unavailable at editing.

In an auto indexing mode there is an automatic filling of that part of string which contains number (after the mark text). Auto indexing is allowed for marks with same text designation. Mark type— with a leader, on a line or without a leader— is of no importance. Also has no value a way of setting of numbers of the previous marks— manually or automatically.

For creation of a mark with automatically assigned number fulfill following operations.

1. Activate the **Auto Numbering** switch on the Property bar. The **Text** field is empty by default. If the option is enabled to **Save the Text** on a tab the **Sign** toolbar, the text placed over a shelf of the last created designation is displayed in the **Text** field.
2. Right-click in the field **Text**. A context menu, containing an alphabetical list of all marks used in the documents, will appear on the screen.



A mark presented with the characters < > doesn't have a text and consists of a number only.

3. Select from the shortcut menu the necessary mark. In the field **Text** the denotation of a mark with following free number assigned to it will be displayed.

It is possible to edit the denotation, to enter the additional text over and under a shelf, to enter the remark in the dialog to enter text (see Section 3.3.103 on p. 1072).



If the mark/ reference designation with automatically assigned number is deleted from the document, numbers of the remained objects do not vary.

You can customize system so that the mode of auto indexing for marks of all types has been included by default. To do that, enable the **Auto Numbering** option in the **Mark/Reference Designation— Common Settings** items in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

Change of a state of the switch **Auto indexing** on the Panel of properties to equivalently change of a state of an option **Auto indexing** in dialog of the common customizations.

If the auto indexing mode is switched off, the field **Text** is empty. In this case number to mark/ reference designation is assigned manually in text entering dialogue.

3.3.13.3.3. Parameters of a Mark/Reference Designation without a Leader

To set up the display style of a mark/reference designation without a leader, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.4.

Tab. 3.3.4. Form of Mark/Reference Designation without Line Control

Element	Description
Form	<p>The list for designation form selection.</p> <p>If the field is empty, the Without Form type is selected, and the designation is created without a form. Expand the Line style list and select necessary line to change the form. The list of forms available for selection, as well as the sequence of forms in the list, are defined by the filter settings set in the Mark/Reference Designation— Without Leader— Form Filter item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).</p> <p>If the type Without Form is selected then customization is not required. If one of the designation forms is selected, the fields for form size input appear on the Property bar: Width, Height, or Size (see Table 3.3.5).</p>
Width	The field to input or select (in the counter) the width of the circumscribing rectangle of the current form.
Height	The field to input or select (in the counter) the height of the circumscribing rectangle of the current form.
Outline Dimension	The field to input or select (in the counter) the size of the current form. Form dimension — the size of the geometrical figure limiting outside the form of the denotation. Form dimension is the side of a square or diameter of a circle — depending on form type.








Tab. 3.3.4. Form of Mark/Reference Designation without Line Control

Element	Description
Style of Line	The field displaying the current form display style. Expand the Line Style list and select necessary line to change the style. The list of styles available for selection, as well as the sequence of styles in the list, are defined by the filter settings set in the Mark/ Reference Designation— Without Leader— Line Filter item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).
By default	If this option is enabled, all current settings on the Options tab of the Property bar will be used while creating the following marks/reference designations until the end of the session. If the option is disabled the setup is only applied to current designation.





* The size of a form doesn't depend on the input text length. If the string appears longer the text field in shape, there will be an automatic contraction of the text.

In Table 3.3.5, the fields for form size input are presented. They appear on the Property bar after form selection. The types of form sizes to specify are presented in parentheses.

Tab. 3.3.5. The Types of Form Sizes for marks/reference designations without leaders


Form	The names of input fields and the size types
Without Form	You don't need to specify sizes
	Circle Size (diameter)
	Rectangle Width (width), Height (height)
	Square Size (square side length)
	Rhomb 1 Width (width of the circumscribed rectangle) Height (height of the circumscribed rectangle)
	Rhomb 2 Size (side length of the circumscribed square)
	Hexagon Size (diameters of the circumscribed circle)
	Triangle 1 Size (diameters of the circumscribed circle)

Tab. 3.3.5. The Types of Form Sizes for marks/reference designations without leaders

Form	The names of input fields and the size types
	Triangle 2 Size (diameters of the circumscribed circle)
	Rounded rectangle Width (distance between the centers of the half-circles) Height (heights)
	Circle with a Vertical Divider Size (diameter)
	Double circle Size (diameter of the inner circle)

The default form sizes and the distance between a divider and a text are set in the **Mark/Reference Designation— Without Leader— Parameters** in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.4. Mark/Reference Designation with Leader


 To create a mark/reference designation with a leader, call the **Mark/reference designation with leader** command.

Specify a start point (**p**) of designation first branching.

Select the start point **p1** of leader bend.

Then, select the start points of other branchings. Their number is not limited.

Input the designation text (see Section 3.3.13.3.1 on p. 1071).

 To do automatic assignment of free numbers to elements while inputting the text, activate the **Auto Numbering** switch. For details about autonumbering see Section 3.3.13.3.2 on p. 1073.



Contextual menu of the **Text** field contains all marks saved in the document. To create mark with already existing text designation you can select it from this menu (Fig. 3.3.101 on p. 1070). It will be assigned sequential number in the mode of autonumbering.

If text entered for the current mark/reference designation (the one being created) shall be used for the rest of the same designations created within this command call, enable the **Save Text** option. If the option is disabled, then each next designation is created without text.

Text can be link source for another text object in this document.

Set up the parameters of marks/reference designations on the **Options** tab of the Property bar (see Section 3.3.13.4.1).

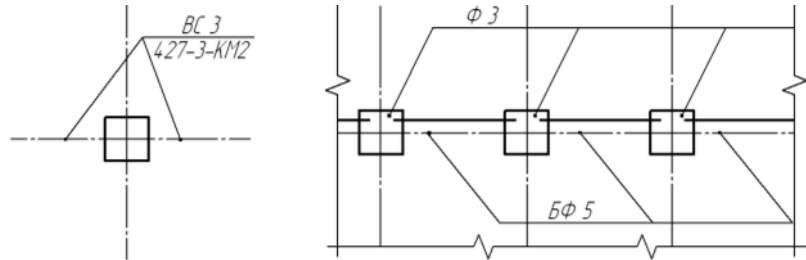


Fig. 3.3.104. Mark/reference designation with leader samples



By default, when creating lines marks/reference designations, the branching mode is on. Pressed **Add Branches** button on Advanced Control panel shows this. In this mode each newly specified point is recognized by the system as the start point of sequential branching.



In the object handle editing mode, of editing of object defining points you may make changes in designation configuration. Adding the branchings is impossible in this mode. To activate the mode of editing click the **Edit Points** button on the Special Control Panel.

Move the pointer to any handle (these points are displayed as small black squares). Shape of cursor will be changed.

Change the position of handles or delete unnecessary points (see Section 3.3.7.4.3 on p. 1027).

You can arbitrary switch between these modes with buttons of the Special Control Panel and commands of the contextual menu. On exiting a mode another one is enabled automatically.



The **Copy Properties** button on the Advanced Control panel allows you to copy text into the current designation from an existing mark/reference designation (of the same or other type).



To fix that, click the **Create Object** button on the Advanced Control Panel.

You can create several designations during one call of the **Mark/Reference Designation with Leader** command. Specify new points **p** and **p1** after you have clicked **Create Object** button




3.3.13.4.1. Parameters of Marks/Reference Designations with a Leader

To set up the display style for marks/reference designations with leaders, activate the **Parameters** tab on the Property bar. The control elements located on it are represented in Table 3.3.6.

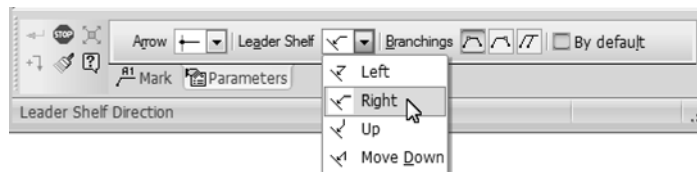
Tab. 3.3.6. Control elements to manage the display style of marks/reference designations with leaders

Element	Description
Arrow	The Arrow list serves to select an arrow type of mark/reference designation. The list of arrows available for selection as well as the sequence of arrows in the list, are define by the filter settings set in the Mark/Reference Designation — With Leader — Arrow Filter in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

Tab. 3.3.6. Control elements to manage the display style of marks/reference designations with leaders

Element	Description
Leader Shelf	<p>The control widget that allows selection of designation shelf location. The Shelf option has different appearance depending on which switch is active in the Branchings.</p> <p>If the switch From Shelf Start or From Shelf End is active, Shelf control represents the revealing list, allowing to select a shelf direction. In the list, select the desired shelf display direction: Left, Right, Top, Bottom (Fig.3.3.105).</p> <p>If the Parallel switch is active, then Shelf control represents an option group, allowing to select shelf position — horizontal or vertical (Fig. 3.3.106)*.</p>
Branchings	<p>The group of radio button that allow to select the method to create branches. Activate the desired switch:</p> <p>From shelf start,</p>  <p>From shelf end,</p>  <p>Parallel.</p> 
By default	<p>If this option is enabled, all current settings on the Options tab of the Property bar will be used while creating the following marks/reference designations until the end of the session.</p> <p>If the option is disabled the setup is only applied to current designation.</p>

* As long as the type of the **Shelf** control element depends on currently active switch in the **Branchings** group, it is recommended to set up leader shelf after the mode of adding branchings has been selected.

Fig. 3.3.105. **Shelf** control element that appears as a drop-down list

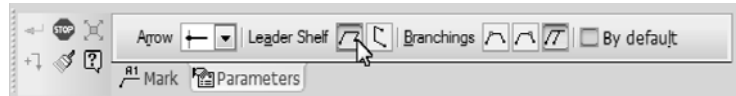


Fig. 3.3.106. **Shelf** control element that appears as a group of switches

The distance between text and shelf, as well as arrow sizes, are set in the **Mark/Reference Designation— With Leader— Parameters** item, and the view of arrows and obliques is set in the **Mark/Reference Designation— With Leader— Arrows and Obliques** in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.5. Mark/Reference Designation on Line



To create a mark/reference designation on a line, call **Mark/Reference Number Designation on Line**.

Specify the cursor the line for denotation allocation. Once the object is highlighted, click the left mouse button.



To replace the object, click the button **Reselect** in Advanced control panel and select a new line.



Ghost mark/reference designation will be displayed.



The group of switches named **Text Location** helps you select the location of the designation: **Above Line**, **On Line**, or **Below Line**. In the course of construction by means of the mouse the switch corresponding to current position of the phantom automatically becomes more active.



Input the designation text (see Section 3.3.13.3.1 on p. 1071).



To do automatic assignment of free numbers to elements while inputting the text, activate the **Auto Numbering** switch. For details about autonumbering see Section 3.3.13.3.2 on p. 1073.



Contextual menu of the **Text** field contains all marks saved in the document. To create mark with already existing text designation you can select it from this menu (Fig. 3.3.101 on p. 1070). It will be assigned sequential number in the mode of autonumbering.

If text entered for the current mark/reference designation (the one being created) shall be used for the rest of the same designations created within this command call, enable the **Save Text** option. If the option is disabled, then each next designation is created without text.

Text can be link source for another text object in this document. For details about links, see Section 4.1.4.3 on p. 1337.

Moving the cursor, select text allocation — over a line, on a line or under a line and left click.

If the specified point does not belong to the selected line, then the symbol position will be determined by the projection (mapping) of the specified point on the line or its extension. In the latter case the line will be automatically continued on the necessary distance by a portable line with style the thin.



Extending NURBS and Bezier curves is not possible.

To precisely position the cursor, use snaps or the Geometric Calculator menu for the **Symbol Location** field on the **Symbol** tab of the Property bar.

The distance between text and line is set in the **Mark/Reference Designation— On Line—Parameters** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

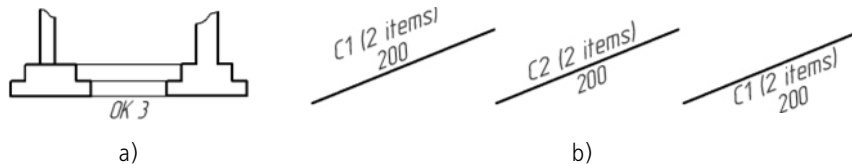


Fig. 3.3.107. Mark/reference designation on line:
a) usage example, b) text location variants



The background under the denotation is always cleared irrespective of the customizations made in dialog of parameters of mapping of overlapped objects.



Button **Copy Properties** on the Advanced Control panel allows you to copy text into the current designation from an existing mark/reference designation (of this or other type).

You can create several designations during one call of the **Mark/Reference Designation on Line** command.

3.3.13.6. Cut line



To create a cut line, call the **Cut Line** command.

Creation of a cut line for civil engineering is generally similar to that for mechanical engineering (see Section 3.3.7.10 on p. 1040).

Differences are as follows:

- ▼ The designation uses numbers instead of letters (see Section 3.3.13.6.1);
- ▼ The **Create View** option controls creation of a new view after the cut line is created. If the option is enabled, the new view creation command is launched automatically after the cut line is created (see Section 3.3.14.1 on p. 1110). After this command is executed, the drawing will have the view with its caption associatively related to the created cut line.

3.3.13.6.1. Creating cut line designation text

Use the **Text** field of the Property Bar to enter cut line designation text. This field displays the automatically created designation text- cut number. You can select the required cut number from the context menu of this field (Fig. 3.3.108).

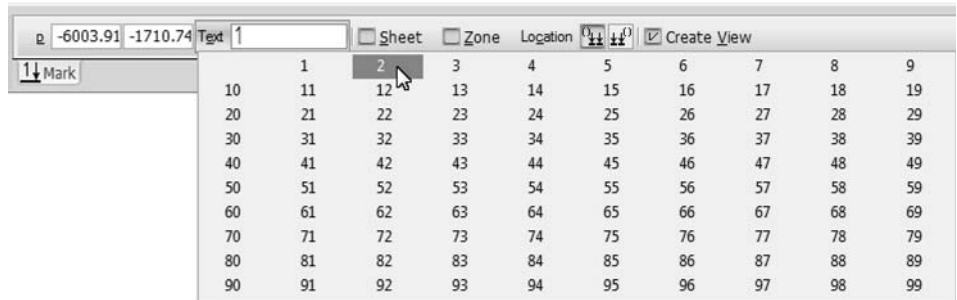


Fig. 3.3.108. Symbol selection menu

If the cut line designation should include a sheet number or a designation symbol of the zone where the view with the respective image will be positioned, enable the **Sheet** or **Zone** option on the Property Bar.

The sheet number or the zone designation symbol is automatically included into the cut line designation after creating the view and generating an associative relation between the caption of this view and the cut line designation (see Section 3.3.14.1 on p. 1110).



You cannot create sheets and views in fragments. Thus, the options **Sheet** and **Area** are not available.

The cut line designation text can be also generated in the designation text input dialog. To call this dialog, click the mouse within the **Text** field or call the **Caption Text...** command from the contextual menu. You can simply start inputting the text; the dialog will automatically appear on the screen.

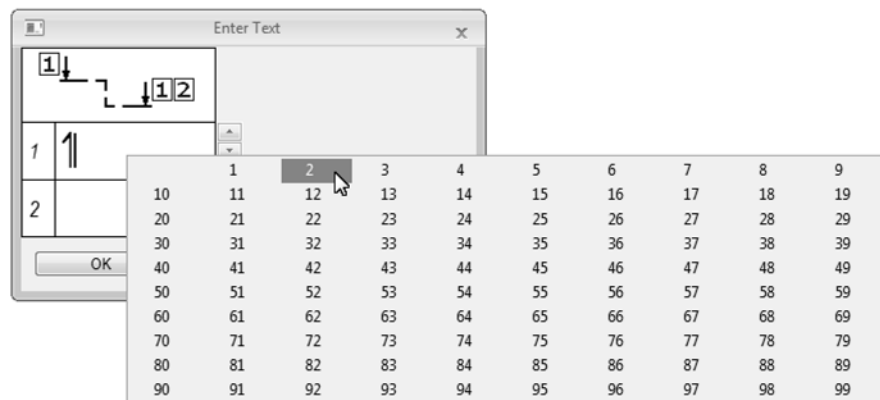


Fig. 3.3.109. The cut line designation text input dialog

The first input field of the dialog is intended for numerical designation of the cut. A double click in this field calls the symbol selection menu (Fig. 3.3.109). The buttons with arrows to the right of the text input field allow to scroll the list of cut numbers in any direction.

In the second field, you can input additional information, for example, area designation.



If the **Sheet** or **Zone** option is enabled on the Property Bar, the second field is unavailable in the designation text input dialog.

If needed, change text parameters (character size, color, etc.) using the controls on the tabs of the Property Bar.

After inputting and formatting the text in the fields, click the **OK** button in the dialog.



If text parameters (font, height, color, etc.) of all cut line designations in a document are different from the current default parameters, it is recommended to set the desired parameters as default instead of setting up every single legend. For this purpose, use the **Designations for constructions — Cut line** subsection of the current document setup dialog (see Section 3.3.13.2 on p. 1069).

3.3.13.7. Node Designation



To create a node point designation, call the **Node Point Designation** command.

There is a base point of the node point designation in the center of the node point outline.

Enter the coordinates of the node point center on the Property bar or specify the point defining its location in the document window.

Input the designation text (see Section 3.3.13.7.1).

Node point designation text can contain a reference to another object (node point number, for example) as well as to be the reference target. For details about links, see Section 4.1.4.3 on p. 1337.

Specify the contour size: for a circle— diameter or radius; for rectangles— width, height, and rounding radius.

Set up the display parameters for node point designations on the **Options** tab of the Property bar (see Section 3.3.13.7.2 on p. 1084).

Option **Create View** allows creating a new view following node creation.

If the option is enabled, the new view creation command is launched automatically after the node is created (see Section 3.3.14.1 on p. 1110). After the execution of this command, a view will appear in the drawing with its designation associatively related to the created node.



You can give up creating a new view by clicking the **Interrupt Command** button. However, we do not recommend doing this, because it will take additional time to manually create the view and build the links between its designation symbol and the node symbol. Besides that, placing node image in a separate view helps quickly change the image scale and makes views placement on the drawing more convenient.

While placing a node, a hyperlink may be created automatically to link the designation with the newly created view (see Section 3.3.14.2 on p. 1111).

Select the start point **p2** of leader bend.

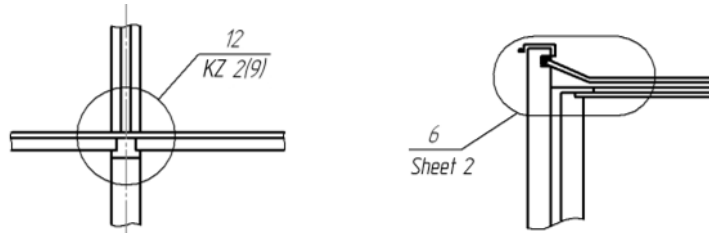


Fig. 3.3.110. Node point designation examples

3.3.13.7.1. Node Point/Sectional Node Point Designation Text Input

To input text while creating or editing a node point/sectional node point designation, use the dialog shown in Fig.3.3.111.

The dialog is called by clicking in the **Text** field on the **Symbol** tab of the Property bar by using the **Text...** command. in the context menu of the drawing field. You can simply start inputting the text; the dialog will automatically appear on the screen.

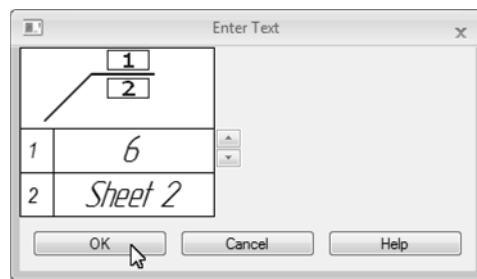


Fig. 3.3.111. General settings of node point and sectional node point designation

By default, the text represents number defined automatically. It is first free number among denotations of knots and knots in the section, available in the current document. The buttons with arrows to the right from field 1 allow scrolling the list of designation numbers in any direction.

Enter the text into the entry fields.

To input text in a different field, press the **<Tab>** key or the **<Shift> + <Tab>** key combination.



To insert a tab while inputting text, press the **<Ctrl> + <Tab>** key combination.

While inputting the text of a node point designation, you can use text templates. To call the **Text Template Library**, double-click in the input field. For detailed information on text templates — see Section 4.1.2.11.6 on p. 1312.

Unless and until the dialog to enter text is displayed on the screen, elements on the Property bar and commands in the Main menu allowing text editing will be available. Some of them can be called from the context menu of the dialog's input field.










If the text parameters (font, height, color etc.) of all node point designations in the document are different from the default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Sectional Node Point Designation—Text** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.7.2. Node Point Designation Parameters

To set up the node point designation display style, activate the **Options** tab of the Property bar. The control elements located on it are represented in Table 3.3.7.

Tab. 3.3.7. Control elements of the node point display style

Element	Description
Form	The list for selection of the contour around a node. In the list that drops down, select a designation form:
	Circle,
	Rectangle
	Rounded Rectangle
Leader Shelf	The list for selection of designation shelf location. In the list that drops down, select the desired leader shelf direction:
	To the left,
	To the right,
	Up,
	Down.
Save Text	If the option is enabled, the text below the shelf (input for the current node point designation) will be suggested as default for the following node point designations until the end of the session. If the option is disabled, each new designation will only contain a text above the shelf.

Tab. 3.3.7. Control elements of the node point display style

Element	Description
By default	If this option is enabled, all current settings of the Options tab on the Property bar will be used to create the following designations until the end of the session. If the option is disabled the setup is only applied to current node point designation.

The distance between text and shelf are given in **Sectional Node Point Designation— Parameters** in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.8. Sectional Node Point Designation,



To create designation sectional node point, call **Sectional Node Point Designation**.

Input the coordinates of the dash anchor point (**p**) on the Property bar or specify its location in the document. The anchor point of a dash is its mid point.

A ghost designation appears on the screen with the default parameters.

Enter length of a dash in the field **Dash**. Select the start point **p1** of leader bend. The corner of an inclination of a dash is calculated automatically and **Corner** is brought in the field. The angle of deviation of the OX-axis of the current coordinate system. If it is necessary to install a certain corner of an inclination of a dash, enter this value from the keyboard in the field **Angle** and fix it.



To start construction from a point of the beginning of a shelf, activate input field of co-ordinates of a point **p1** on the Panel of properties, having pressed the switch, and specify points (**p1**) and (**p**).

Set up the sectional node designation parameters on the Options tab of the **Property** bar (see Section 3.3.13.8.1 on p. 1086).

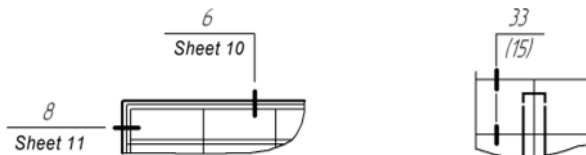


Fig. 3.3.112. Sectional node point designation samples

Input the designation text (see Section 3.3.13.7.1 on p. 1083).

The text of the denotation of knot in section can contain the reference to other object, for example, knot number. For details about references, see Section 4.1.4.3 on p. 1337.



By default, when creating lines sectional node point number designation, the dash adding mode is on. In this case, the **Add Dashes** button on the Advanced Control panel is pressed. In this mode each newly specified point is recognized by the system as the midpoint of sequential

dash. You can create some dashes successively on a line-leaders, specifying their anchors. If dashes have different length, enter in the field **Dash** the size of each following dash.



In the object handle editing mode, you can change the dash inclination angle, the location of the dashes and the shelf, as well as delete the dashes. To activate the mode of editing click the **Edit Points** button on the Special Control Panel.

Move the pointer to any handle (these points are displayed as small black squares). Shape of cursor will be changed.

Change the position of handles or delete unnecessary points (see Section 3.3.7.4.3 on p. 1027).



To change the dash inclination angle in the handle editing mode, unlock the **Angle** field.

You can arbitrary switch between these modes with buttons of the Special Control Panel and commands of the contextual menu. On exiting a mode another one is enabled automatically.

Option **Create View** allows creating a new view following the sectional node creation. This view becomes current and you can proceed with drawing the image in it.







To fix that, click the **Create Object** button on the Advanced Control Panel.

You can create a number of designations for one call of the **Sectional Node Point Designation** command. Specify new points **p** and **p1** after you have clicked **Create Object** button

3.3.13.8.1. Sectional Node Point Designation Parameters

To set up the display style of sectional node point designations, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.8.

Tab. 3.3.8. Control elements of the node point display style

Element	Description
Leader Shelf	The list for selection of shelf direction relative to its start point (p1). Specify the direction in the list that drops down:
	Left,
	Right,
	Up,
	Down.

Tab. 3.3.8. Control elements of the node point display style

Element	Description
Save Text	If the option is enabled, the text designation of the current node point will be suggested as default for the rest of the designations of this type created during this command call. If the option is disabled, then each next designation is created without text below the shelf.
By default	If this option is enabled, all current settings of the Options tab on the Property bar will be used to create the following designations until the end of the session. If the option is disabled, the setting works for the designation only.

The line style of dashes and line between them, as well as the display of lines between dashes, are defined by the settings in the **Sectional Node Point Designation — Common Settings** item; and the default dash length and the gap between dash and leader are defined in the **Sectional Node Point Designation — Parameters** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.9. Node Point Number



To create a node point number designation, call the **Node Point Number** command.

After calling the command the ghost of node point number is displayed. There is a snapping point at the center of the circle.

Enter node point number snap point (p) coordinates on the Property bar or specify the point defining node point number location in the document window.

Input the designation text (see Section 3.3.13.9.1).



Fig. 3.3.113. Examples of node point number designations



To fix designation, click the **Create Object** button on the Advanced Control Panel.

The designation form type and the sizes are specified in the **Node Point Number—Parameters** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

You can create a number of designations during one call of **Node Point Number** command. Specify new location point of the node point number after clicking **Create Object** button to this effect.



Node point number image created with **Node Point Number** command has the following advantages as compared to the same image created by geometrical constructions:

- ▼ Keeps sizes and positional relationship when rotating, scaling or symmetry transformation.
- ▼ Can contain reference to another object (node point designation, for example) as well as to be the reference.

3.3.13.9.1. Text into Node Point Number Designations

To input text into a designation while creating and editing a node point number, use the dialog shown in Fig.3.3.114.

The dialog is called by clicking in the **Text** field on the Property bar or by calling the **Text...** command in the context menu of the drawing field. You can simply start inputting the text; the dialog will automatically appear on the screen.

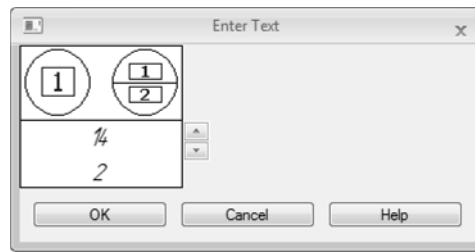


Fig. 3.3.114. Dialog to enter node point number text

Enter the node point number text in the entry field. A separator (a horizontal line in the middle of the shape) will be automatically created, if a two-rowed text is entered. The first text row will be arranged above the separator and the second one— below. Press the **<Enter>** key to create a new row or divide the existing one into two rows.

The buttons with arrows to the right from the input field allow you to scroll the list of designation numbers in any direction.

The text will be automatically narrowed if the entered row length will be larger than designation circle diameter.

Unless and until the dialog to enter text is displayed on the screen, elements on the Property bar and commands in the Main menu allowing text editing will be available. Some of them can be called from the context menu of the dialog's input field.



If the text parameters (font, height, color etc.) Of all node point number designations in the current document are different from the default values, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Node Point Number—Text** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).



To create a reference to a node point designation in a node point number, create this designation preliminarily using the **Node Point Designation** command or the **Sectional Node Point Designation**. Reference insertion command can be called from **Insert** menu or from the dialog entry field contextual menu. Reference target should be in the same document as the node point number. For details about links, see Section 4.1.4.3 on p. 1337.

3.3.13.10. Brace



To create a brace used for fragment designations, call the **Brace** command.

Specify the first anchor point (**p1**) and the second one (**p2**), if they are known. Brace length and slope will be defined automatically.

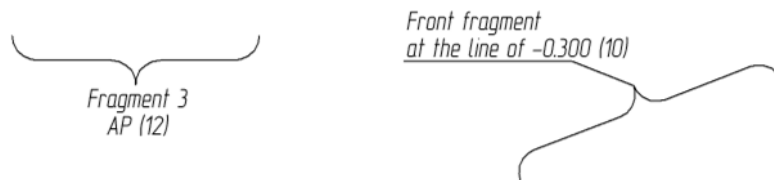


Fig. 3.3.115. Brace samples

If the first brace anchor point, brace length and slope are known than specify these parameters soever in any order. Note that the minimal brace length depends on the rounding radius (see Section 3.3.13.10.2 on p. 1090).

Brace ghost image will be displayed on the screen.

You should specify shelf start point (**p3**) if brace text is arranged on the leader shelf.



Group of switches named **Type** allows you to select the brace orientation. horizontal, vertical, or arbitrary.



Group of switches named **Direction** allows you to select a brace direction.



The default brace direction is set in the **Brace— Common Settings** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

Input the designation text (see Section 3.3.13.10.1 on p. 1090).

Brace text can contain a reference to another object or can be a target for the reference. For details about links, see Section 4.1.4.3 on p. 1337.

Set up the brace parameters on the **Options** tab of the Property bar (see Section 3.3.13.10.2 on p. 1090).



Button **Edit Points** on the Advanced Control panel is used to go into the leader handle editing mode. You can create jogs in the leader and move its characteristic points with mouse (see Section 3.3.7.4.3 on p. 1027). **Edit Points** button is available if one of the types of arrangement text on the shelf has been selected and the shelf start point (**p3**) has been specified.

Some features of the brace editing process are described in Section 3.3.13.10.3 on p. 1092.



To fix that, click the **Create Object** button on the Advanced Control Panel.

You can create a number of designations during one call of **Brace** command. Specify new anchor points after you have clicked **Create Object** button.

3.3.13.10.1. Brace Designation Text Input

To input text while creating or editing a brace designation, use the dialog shown in Fig. 3.3.116. The dialog is called by clicking in the **Text** field on the Property bar or by calling the **Text...** command in the context menu of the drawing field. You can simply start inputting the text; the dialog will automatically appear on the screen.

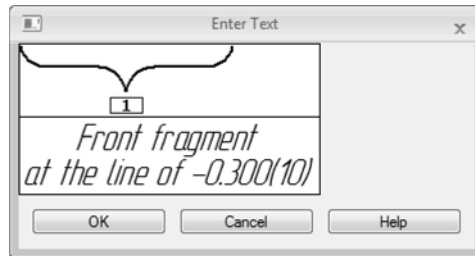


Fig. 3.3.116. The brace text input dialog

Enter the text of the denotation in the field of input.

To input several lines, use the **<Enter>** key to start a new line.



To insert a tab while inputting text, press the **<Ctrl> + <Tab>** key combination.

A double click in the input field allows you to insert a text template (see Section 4.1.2.11.6 on p. 1312).

Unless and until the dialog to enter text is displayed on the screen, elements on the Property bar and commands in the Main menu allowing text editing will be available. Some of them can be called from the context menu of the dialog's input field.



If the text parameters (font, height, color etc.) of all fragment (brace) designations in the document are different from the current default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Brace—Text** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).






3.3.13.10.2. Brace Parameters

To set up the brace display style, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.9.

Tab. 3.3.9. Control elements to manage the brace display style

Element	Description
Text Location	The list for selection of a brace text location method. Select the appropriate variant in the list that drops down.

Tab. 3.3.9. Control elements to manage the brace display style

Element	Description
	While selecting one of the text location variants: On Leader Shelf Right,
	On Leader Shelf Left,
	On Leader Shelf Top,
	On Leader Shelf Bottom a leader with a shelf in the specified direction will be created. The user needs to additionally specify the shelf start point (p3).
	If you select the Auto method, the text will be placed on the outer side of the brace without a leader. The text lines are parallel to the brace. The text is automatically centered relative to the brace.
Radius rounding	After selecting the brace rounding radius. You can input radius values from the keyboard. If you specify a rounding radius for which the brace length is less than the minimal value (see below), the brace length will automatically be assigned the minimal value.
Style of Line	The field to display the current brace line style. Expand the Line Style list and select necessary line to change the style. The list of styles available for selection, as well as the sequence of styles in the list, are defined by the line filter settings in the Brace — Line Filter item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).
Save Text	If the option is enabled, the current brace text designation will be suggested as default for the following braces until the end of the session.
By default	If this option is enabled, all current settings will be used to create the following braces until the end of the session. If the option is disabled the setup is only applied to current brace.

Minimal brace length is calculated by the following formula

$L=4 \cdot R/M$, where

R — the brace rounding radius,

M — the scale of a view, where the brace is located.

If a brace is created in a view with scale 1:1, or in the fragment, then $L=4 \cdot R$.

The default text location method, automatic or on a shelf is defined by the settings in the **Brace— Common Settings** item, and the default rounding radius, as well as the distance between text and brace, are defined in the **Brace— Parameters** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.10.3. The Features of Brace Editing with the Mouse

The brace has four or five (the number depends on selected arrangement type) handles.

Editing braces by dragging their handles has the following features.

- ▼ Change brace direction is reached by moving point (**p3**) in the center of the brace (Fig. 3.3.117).

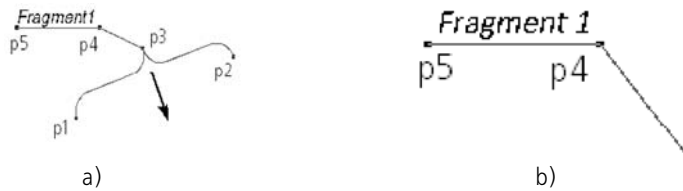


Fig. 3.3.117. Brace direction change:
a) initial state and direction of (**p3**) point move, b) the result of editing

- ▼ When one of anchor points (**p1** or **p2**) moves, brace length changes and the brace rotates around another anchor point. At the same time brace text rotates with the brace, and the text arranged on the shelf is permanent. When moving anchor points, you should note that they cannot be brought together for a length that will be less than minimal brace length (see Section 3.3.13.10.2 on p. 1090). The minimal length depends on the rounding radius (see Table 3.3.9 on p. 1090).
- ▼ Text arrangement type is changed by moving **p4** point (Fig. 3.3.118).

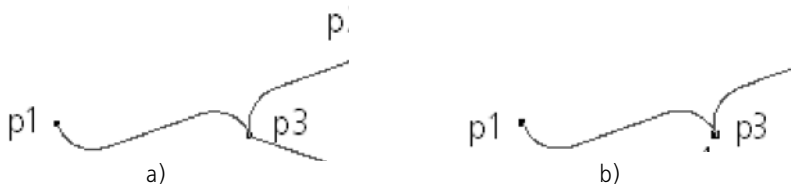


Fig. 3.3.118. Changing the text on a shelf to a designation text:
a) initial state and direction of (**p4**) point move, b) the result of editing

- ▼ While moving the handles, the horizontal/vertical orientation of a brace doesn't change being specified with the **Type** switch group.

3.3.13.11. Leader Text



To create the leader text for multi-layer construction designations, call the **Leader Text** command.

Specify a start point (**p**) of leader text first branching.

Select the start point **p1** of leader bend.

Then, select the start points of other branchings. Their number is not limited.

Input your leader text (see Chapter 3.3.13.11.1 on p. 1093).

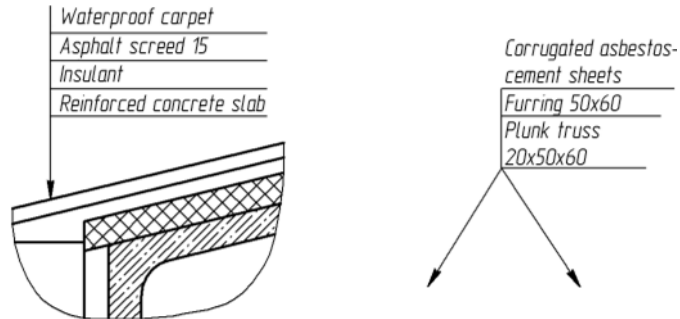


Fig. 3.3.119. Examples of leader texts

Set up the leader text parameters on the **Options** tab of the Property bar (see Section 3.3.13.11.2 on p. 1095).



By default, the branching mode is on while creating a leader text. Pressed **Add Branches** button on Advanced Control panel shows this. In this mode each newly specified point is recognized by the system as the start point of sequential branching.



In the object handle editing mode, of editing of object defining points you may make changes in designation configuration. Adding the branchings is impossible in this mode. To activate the mode of editing click the **Edit Points** button on the Special Control Panel.

Move the pointer to any handle (these points are displayed as small black squares). Shape of cursor will be changed.

Change the position of handles or delete unnecessary points (see Section 3.3.7.4.3 on p. 1027).

You can arbitrary switch between these modes with buttons of the Special Control Panel and commands of the contextual menu. On exiting a mode another one is enabled automatically.



To fix that, click the **Create Object** button on the Advanced Control Panel.

You can create a number of designations during one call of the **Leader Text** command. Specify new points **p** and **p1** after you have clicked **Create Object** button

3.3.13.11.1. Leader Text Input

To input the designation text while creating or editing a leader text, use the dialog shown in Fig. 3.3.120.

The dialog is called by clicking in the **Text** field on the Property bar or by calling the **Text...** command in the context menu of the drawing field. You can simply start inputting the text; the dialog will automatically appear on the screen.

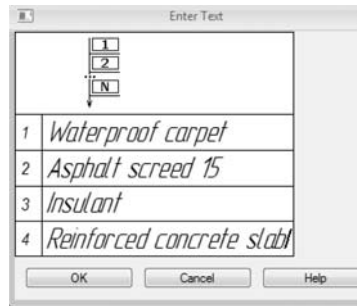


Fig. 3.3.120. Enter leader text dialog

The table is displayed in the dialog window. It contains two columns. There are record numbers in the left column. There is a field to enter the text in the right column.

Enter the text into the entry field.

To jump to next leader shelf entry — press `<Shift> + <Enter>` key combination or the `<Tab>` key. New row with next leader shelf number will appear in the dialog window.

To input several lines on the same shelf, use the `<Enter>` key to start a new line.



To insert a tab while inputting text, press the `<Ctrl> + <Tab>` key combination.

You can use text templates while inputting a leader text. To call the **Text Template Library**, double-click in the input field. For detailed information on text templates — see Section 4.1.2.11.6 on p. 1312.



You can insert several text templates into a leader text at the same time. At the same time the number of additional rows (and, respectively, leader shelves in the leader text) created in the dialog will be equal to the number of selected templates in the **Librarian** window.

Unless and until the dialog to enter text is displayed on the screen, elements on the Property bar and commands in the Main menu allowing text editing will be available. Some of them can be called from the context menu of the dialog's input field.



If the text parameters (font, height, color etc.) of all leader text designations are different from the current default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Leader Text — Text item** in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).










You can create a link to another text object in the same document. The command to insert a link can be called from the **Insert** menu or from the contextual menu of the entry field. For details about links, see Section 4.1.4.3 on p. 1337.

Upon finishing text enter click the **OK** button.

3.3.13.11.2. Leader Text Parameters

To set up the leader text display style, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.10.

Tab. 3.3.10. Control elements to manage the leader text display style

Element	Description
	Arrow The list for selection of a leader text arrow type. The list of arrows available for selection, as well as the sequence of arrows in the list, are defined by the filter settings in the Leader Text — Arrow Filter item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).
	Direction Shelves This group of toggles allows selecting shelf direction of the leader text: Leader Shelves Right or Leader Shelves Left .
	Text Up/Down This group of switches allows selecting the arrangement of leader shelves with text relative to p1 point. If the Text Down switch is active, the beginning of the top shelf will be matched with the p1 point; if the Text Up switch is active, the beginning of the top shelf will be matched with the p1 point. While changing shelf locations, their sequence doesn't change.
	
	Shape Type A group of toggles allows selecting the form of the leader text top shelf. If the Type 1 switch is active, the top shelf will be displayed with an additional vertical segment. If the Type 2 switch is active, the top shelf will be displayed without additional items.
	
	Alignment This group of toggles allows selecting the mode of leader shelf length alignment. If the Constant Leader shelf Length switch is active, all leader shelves have the same length equal to the longest shelf's length. Its length is composed of its text row length and offset values. If the Shelves by Text Length switch is active, shelves can be of different lengths. The length of each shelf is composed of its text row length and offset values.
	
	Save Text If the option is enabled, the current text will be suggested as default for the following leader text until the end of the session. If the option is disabled, then each next leader line is created without text.

Tab. 3.3.10. Control elements to manage the leader text display style

Element	Description
By default	If this option is enabled, all current settings will be used to create the following leader texts until the end of the sessions. If the option is disabled the setup is only applied to current leader text. The default shelf form and the alignment method are defined in the Leader Text — Common Settings item.

The default shelf form and the alignment method are defined in the **Leader Text — Common Settings** item. The distance between text and shelf and the arrow sizes are defined in the **Leader Text— Parameters** item. The arrow and oblique type is defined in the **Leader Text— Arrows and Obliques** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.12. Straight Coordination Axis



To create straight coordination axis mark, call the **Straight Coordination Axis** command. Specify an axis start point (**p1**). A ghost designation appears on the screen with the default parameters.

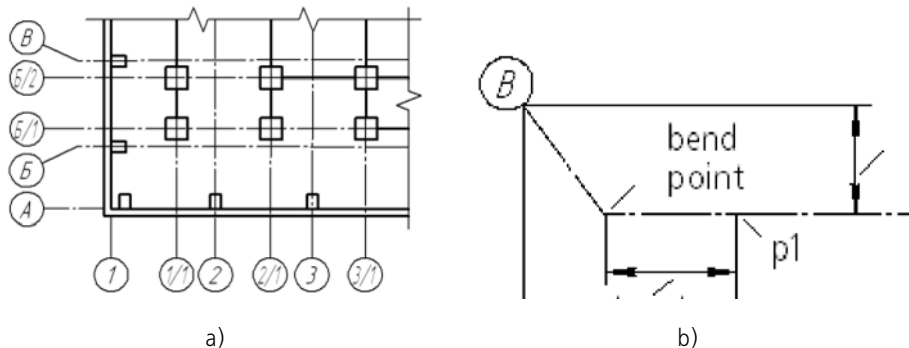


Fig. 3.3.121. Strait coordination axis:
a) usage example, b) axis parameters

Enter the symbol text.



To do automatic number Assignment while inputting the mark text, activate the **Auto Continue** switch (see Section 3.3.13.12.1 on p. 1097).

For details about coordination axis designation creation, see Section 3.3.13.12.2 on p. 1098.

Specify an axis end point (**p2**).

In the document the direct coordination axis exiting for points **p1** and **p2** on the values of hanging indents will be created. Thus, length of a segment between points **p1**, **p2** and an axis corner **the Length** and **the Corner** will be defined automatically and brought in fields.

While creating an axis, you can specify its parameters on the **Parameters**, **Jut1**, and **Jut 2** tabs of the Property bar (see Section 3.3.13.12.3 on p. 1099 and Section 3.3.13.12.4 on p. 1101).

Calling the **Straight Coordination Axis** command once, you can create several coordination axes.



While creating axes with the same parameters (all or a certain group), it is convenient to use the **Remember Status** command. An example of coordination axis creation is given in Section 3.3.13.15 on p. 1108.

A coordination axis can have additional designations (Fig. 3.3.122). The sequence of their creation is described in Section 3.3.13.12.5 on p. 1102.

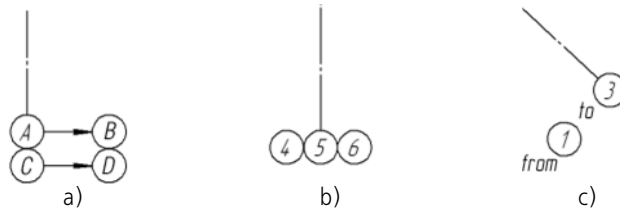


Fig. 3.3.122. Elements of additional designations:
a) orientation pointer, b) mark, c) text

3.3.13.12.1. Auto Continue



The **Auto Continue** switch allows you to automatically assign a number or a letter to the coordination axis you are creating. The switch is accessible at mark/reference designation creation and is hidden at editing.

Auto continuation operates for the **axis denotation** — characters or the numbers entered into the first input field of dialogue of text entering (see Fig. 3.3.124 on p. 1099) or selected in the context menu of the **Text** field on the Property bar (see Fig. 3.3.123).

- ▼ If the previous axis has the digital denotation (number), the following axis following number in ascending order is assigned.
- ▼ If the previous axis has a lettering the following axis the character following one after another is assigned. The sequence of letters is defined by the list specified in the **Coordination Axes — Common Settings** item in the settings dialog of the current document (see Section 9.2.6.14.1 on p. 1798). The letters that are not on the list are ignored while performing auto continue.
- ▼ If the previous axis has the letter and digital denotation, auto continuation operates for the first character. For example, after an axis with denotation **A1** axis **B1** will be created.

If the coordination axis with automatically assigned number or the character is deleted from the document, numbers and characters of the remained axes do not vary.



Auto continuation operates only for the main marks, i.e., the marks attached with coordination axes. Auto continuation is not made for brands of additional denotations.



Auto continuation operates only during one call of the command of construction of axes. That auto continuation operated till the end of session KOMPAS-3D, it is necessary to additionally enable the **Save Text** option on the **Parameters** tab of the Property bar.

If the auto continue mode is off, the coordination axis designation is to be selected manually in the context menu of the **Text** field or specified in the text input dialog.

3.3.13.12.2. Designation Creation

In the **Text** field on the Property Bar the text is displayed that is suggested by the system as the Coordination axis' symbol.

If the field the **Text** is empty, the denotation of a coordination axis is absent. A designation text can be input using two methods.

▼ Method 1

If the denotation contains only number or the character, the **Text** is more convenient to generate it by means of the shortcut menu of weeding. In the menu there are following commands: **Number**, **Character** and **Add "c" Subscript** (Fig.3.3.123).

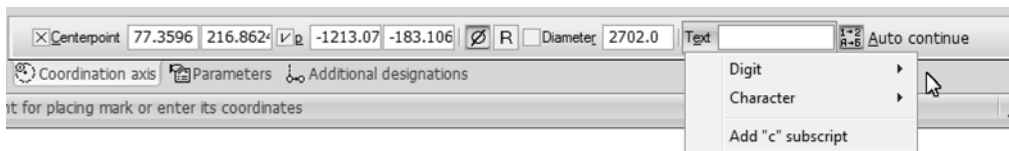


Fig. 3.3.123. The context menu of the **Text** field of a coordination axis.

To create a mark designation, select the desired number or letter in the submenu of the menus **Number** or **Character**.

To add an index to a designation, call the **Add "c" Index** command.

The selected designation will be displayed in the **Text** field.

The list of letters available in the **Character** submenu, as well as the sequence of letters in the list are defined by the list specified in the **Coordinate Axes – Common Settings** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069). Doubled letters are not included in the list— they are formed automatically after the last letter from the list has been used.

▼ Method 2

If you need to input a letter or a number, or edit the current designation, call the text input dialog (Fig.3.3.124) by left-clicking in the **Text** field or by calling the **Text...** command. in the context menu of the drawing field.

The axis designation text structure is shown in the dialog.

If the designation (a letter or a number) has been selected in the submenu, it is located in the first field of the dialog, and the "c" index is located in the third field.

Input or edit the text taking the following information into account.

The part of the designation that intends the use of auto continue is input into the first field (see Section 3.3.13.12.1). You can select a number or a letter by double-clicking in the first field. On the screen, you will see a menu containing the **Number** and **Character** commands.

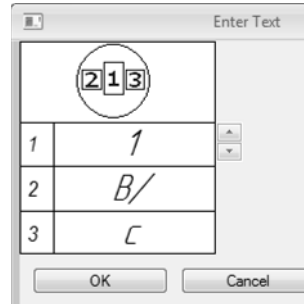


Fig. 3.3.124. The coordination axis text input dialog

The buttons with arrows to the right of the input field allow you to scroll the designation list in any direction. After the single characters dual ones are offered.

If the character and number in the denotation that its part which costs on the first place will vary have been entered into the first string of dialogue of text entering. In case a double letter and a number have been input, the following double letter and the same number are suggested.

In the third field, you can input a text. It will not change while applying auto continue. A double click in the input field allows you to insert a text template (see Section 4.1.2.11.6 on p. 1312).

Unless and until the dialog to enter text is displayed on the screen, elements on the Property bar and commands in the Main menu allowing text editing will be available. Some of them can be called from the context menu of the dialog's input field.





If the text parameters (font, height, color etc.) of all coordination axis designations are different from the current default parameters, it is recommended to set the desired parameters as default instead of setting up every single text. To do that, use the **Coordinate Axes — Text** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.12.3. Display Style Parameters

To set up the coordination axis display parameters, activate the **Options** tab on the Property bar. The control elements located on it are represented in Table 3.3.11.

Tab. 3.3.11. Control elements to manage the coordination axis display style parameters.

Element	Description
	Mark Type The group of radio buttons that allows selection of a mark display style. Activate the Circle or Double Circle switch. For direct and arc axes a brand of the set type is displayed from both ends of a coordination axis.
	

Tab. 3.3.11. Control elements to manage the coordination axis display style parameters.

Element	Description
Dimension	<p>The field of a choice of diameter of a circle of the denotation of a mark of coordination axes.</p> <p>It is also possible to enter the diameter value by typing on the keyboard. Entered values will be added to the list and saved for the duration of a KOMPAS-3D working session.</p> <p>If a double circle is displayed, the specified size defines the diameter of the outer circle.</p>
Dot	The field for input or setting by counter of length of a dotted line.
Spacing	The field to input or select the space length between a dotted line and a dash.
Auto Determination	<p>The option, allowing to select a way of displaying a dash-dotted line of a coordination axis.</p> <p>If the option Auto Determination is included, the length of a dash is calculated automatically with observance of following conditions:</p> <ul style="list-style-type: none"> ▼ The length of a dash is in a range 5...30 mm ▼ Coordination axes start and ended by dashes ▼ Initial and finite points of an axis (p1 and p2) are allocated on dashes. <p>If the Auto Determination is disabled, an axis is created with the dash length less or equal to the value in the Dash field. Axis creation follows the rule saying that the ends of an axis should be dashed.</p>
Dash	The field for input or the job the counter of maximum length of dashes of a coordination axis. This field is accessible when the Auto Determination option is disabled.
Save Text	If the option to Save Text is included, the denotation, the text to and the text after, set for a current axis, will be offered by default for other coordination axes of the same type till the end of a session.
By default	<p>Using this option means all current settings in the Options tab will be used when creating new objects of this type through the end of the working session.</p> <p>If the option is disabled, the settings are applied to the current coordination axis only.</p>

The default diameter and ring width values are set in the **Coordinate Axes— Common Settings** item, and the selection of the dash style and the default dash-dot line parameters, is performed in the **Coordinate Axes— Parameters** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.12.4. Juts

The settings of coordination axis juts are available on the **Jut1** and **Jut 2** tabs on the Property bar.

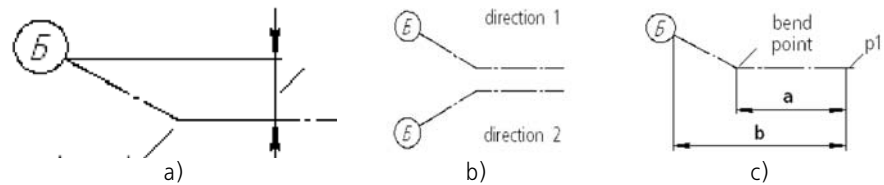





Fig. 3.3.125. Coordinate axis jut parameters:
a) offset, b) direction, c) bend offset

In Table 3.3.12, Control elements of the **Jut 1** tab are described. They allow you to set up juts beyond the start point. The settings of juts beyond the end point are available on the **Jut 2** tab.

Tab. 3.3.12. Control elements to manage coordination axis juts

Element	Description
Hanging	The field for input of the value of a hanging indent for an index point of a coordination axis. Enter the necessary value or set its counter. Input of zero value means absence of a hanging indent.
 Mark	The switch, allowing including or switching off mark displaying in the beginning of a coordination axis. Activate the switch to display axes with marks at the ends of their juts*. If the switch is not active, the coordination axis is displayed without a mark on the end of a hanging indent.
Offset	The field to specify the value of mark offset from a coordination axis (see Fig. 3.3.125, a). Input or select (in the counter) the distance between mark and coordination axis. Input a zero if the mark is located on the axis. The field is available if the Mark switch is active.
  Direction	The group of switches that allows selection of a mark offset direction (see Fig. 3.3.125, b). To change a direction of offset of a brand concerning an axis, make active the switch the Direction 1 or the Direction 2 . The switches are available if the Mark switch is active.

Tab. 3.3.12. Control elements to manage coordination axis juts

Element	Description
Offset of Bend, %	<p>The field to specify the value that defines the bend point location relative to the start point on an axis (see Fig.3.3.125, c).</p> <p>In the field Offset of a Break, % enter or set the counter the ratio of distance from a point p1 or p2 to a point of a break of a hanging indent to all value of a hanging indent. The ratio is set in percentage: $C = (a/b) \cdot 100$, where a, mm — distances between points p1 or p2 and the bend point; b (mm) is the jut length; C (%) is the bend offset.</p> <p>The field is available if the Mark switch is active.</p>
By default	<p>If this option is enabled, all current settings on the Jut 1 tab of the Property bar will be used to create juts beyond the start points of the following axis designations of this type until the end of the session.</p> <p>If the option is disabled, the settings are applied to the current coordination axis only.</p>

* To create chains of marks, go to the **Additional Designations** tab (see Section3.3.13.12.6 on p. 1103).

The default jut and offset values are specified in the **Coordinate Axes—Parameters** item in the settings dialog of the current document (see Section3.3.13.2 on p. 1069).

3.3.13.12.5. Additional Designations

Units of additional denotations of a coordination axis are:

- ▼ Pointers of orientation of an axis
- ▼ Chains of marks
- ▼ Text denotations.

Additional denotations join the main brand. Main Mark is a brand allocated on an axis. If the axis has no main marks, creation of additional denotations is impossible.

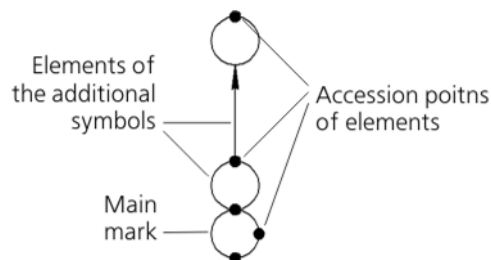


Fig. 3.3.126. The elements of additional axis designations and their connection points.

Additional denotations can be created both in the course of construction, and in edit mode of an axis.

To create additional denotations in the course of axis construction, fulfil following operations.



1. Disable automatic object creation by releasing the **Object Auto Create** button on the Advanced Control panel.
2. Construct the denotation of a coordination axis with the main brand and enter the denotation text
3. Go to the **Additional Designations** tab on the Property bar (see Section 3.3.13.12.6). In group **Units make** active one of switches.
 - ▼ The switch **Add Pointer of Orientation of Axis** allows to add the pointer of orientation of an axis. The axis pointer is added together with a mark.
 - ▼ The switch **Add Mark** allows to add a mark.
 - ▼ The switch **Add Text** allows to add the text denotation in the form of an inscription.

After activation of the switch from the Elements group an element ghost will appear on the screen and all points of possible joining of marks 3.3.126 on an axis will be designated by black points (see Fig.).

The unit the **Pointer of Orientation of Axis** is under construction on a normal to a coordination axis with an arrow routed from the main mark.

4. Customize parameters **Type of Mark** and **Length**.
5. Input the text of the designation you are creating (see Section 3.3.13.12.7 on p. 1104).
6. Specify a point of joining of a unit the mouse. When the point highlights, click the left mouse button.
7. To connect the following unit, make active the necessary switch in group **Units**.
8. To do this, click the **Create Object** button on the Advanced Control Panel.

To add additional denotations to already existing axis, enter into a mode of its editing. For this purpose double click the mouse under the axis denotation. The denotation will change colour. Further fulfill the operations described above in p. 3 — 8.






To edit or delete additional denotations, enter into edit mode and pass to a tab **Additional denotations**. Click under the additional denotation the mouse. The unit will change color. Enter new parameters or delete the additional denotation.

3.3.13.12.6. The Display Style Parameters of Additional Designations

To set up the display style parameters for coordinate axis additional designations (straight, arc, circular), go to the **Additional Designations** tab on the Property bar.

- ▼ While creating an axis, the elements of the **Additional Designations** tab are available if auto creation is disabled and the following points are specified:
 - ▼ **p1** and **2** — for a direct axis,
 - ▼ **pC**, **p1** and **p2** or **pC**, **angle 1**, **angle 2** and **radius** — for arched axis,
 - ▼ **pC** and **p** — for a circle axis.
- ▼ In the course of editing tab units **Additional Denotations** are accessible always. Control elements on this tab are described in Table 3.3.13.

Tab. 3.3.13. Control elements to manage the display style of additional designations

Element	Description
Elements	The group of radio buttons that allows selection of the appearance of the additional elements.
	▼ To add the pointer of orientation of an axis together with a brand, make active the switch to Add Pointer of Orientation of Axis .
	▼ To add a brand, make active the switch Add Mark .
	▼ To add the text denotation in the form of an inscription, make active the switch Add Text .
	To add the next element, you need to activate the switch in the Elements group again.
Mark Type	The group of radio buttons that allows selection of a display style variant for the mark you are adding. Activate the Circle or Double Circle switch.
	Switches Mark Type are accessible, if the switch is active to Add Pointer of Orientation of Axis or Add Mark .
	
Pointer Length	The field to input the axis orientation pointer length. Input it or select in the counter. The field is available if the Add Pointer of Orientation of Axis switch is active.
Text	The field with the text of the current additional designation. The text input features are described in Section 3.3.13.12.7 on p. 1104.

The arrow parameters, as well as the default arrow length, are defined by the values in the **Coordinate Axes— Orientation Marker** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

3.3.13.12.7. Additional Designation Text Input

To input the designation text while creating and editing additional designations:

- ▼ the pointer of orientation of an axis
- ▼ Mark
- ▼ text

use the text input dialog shown in Fig. 3.3.127.

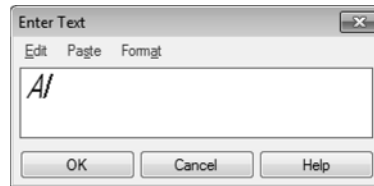


Fig. 3.3.127. The text input dialog for coordination axis additional designations

At addition of units the **Pointer of Orientation of Axis** and **Mark** for dialogue call are required to click the mouse in the field the **Text**.

At addition of a unit the **Text** dialogue appears automatically after the point of joining of the denotation is specified.

Enter the text of the denotation in the field of input.

The text will be automatically narrowed if the entered row length will be larger than mark circle diameter.

Dialog menu contains commands for editing, inserting and formatting text. Some of them can be called from the context menu of the dialog's input field.

Upon finishing text enter click the **OK** button.

3.3.13.13. Coordination Arc Axis



To create a geometric tolerance coordination arc axis, call the **Coordination Arc Axis** command.

Specify a point of center of an arc axes (**pC**). A ghost designation appears on the screen with the default parameters.

Enter the symbol text.



To do automatic number Assignment while inputting the mark text, activate the **Auto Continue** switch (see Section 3.3.13.12.1 on p. 1097).

For details about coordination axis designation creation, see Section 3.3.13.12.2 on p. 1098.

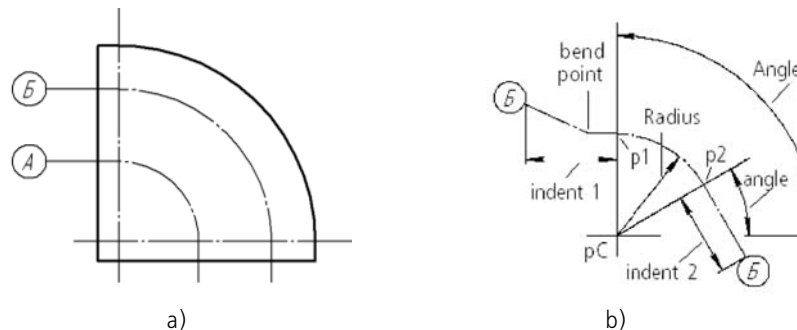


Fig. 3.3.128. Arc coordination axis:
a) usage example, b) axis parameters

Specify initial (**p1**) and finite (**p2**) points of an axis or enter polar co-ordinates of these points into fields **Angle1**, **Angle 2** and **Radius** (or **Diameter**).



By default, the system awaits the radius value as you create a circle. In the **Parameter** group, the **Radius** switch is active.



To enter the diameter, activate the **Diameter** switch.

Angle values are counted from the OX axis of the current coordinate system.

In case an axis is constructed using the mouse, the angles and the arc radius will be defined automatically and entered in the fields **Angle 1**, **Angle 2** and in the current parameter field **Radius** or **Diameter**.



The **Direction** switch group on the Property Bar controls the direction of the arc creation coordinate axis— **counterclockwise** and **clockwise**.

At construction by the mouse after instructions of center (**pC**) and an index point (**p1**) the switch corresponding to current position of the phantom, becomes more active automatically.

To change a direction, it is necessary to spend the cursor along an axis through an index point (**p1**) or to make active the opposite direction switch.

If coordinates of the start and end axis points are entered from the keyboard, the **Direction** group of switches becomes accessible after input of values in any of the three fields: **Centre**, **Angle 1**, **Angle 2** or **Radius** (or **Diameter**).

In the document the arc coordination axis consisting of an arc, limited to points **p1** and **p2**, and rectilinear hanging indents will be created. Hanging indents are allocated on tangents to an axis arc.

Set up the parameters of the coordination arc axis on the **Parameters**, **Jun1**, and **Jun 2** tabs of the Property bar. Arc and straight axes have the same parameters described in Sections 3.3.13.12.3 on p. 1099 and 3.3.13.12.4 on p. 1101.

You can create a number of designations for one call of the **Arc Coordination Axis** command.

Calling the **Straight Coordination Axis** command once, you can create several coordination axes.



While creating axes with the same parameters (all or a certain group), it is convenient to use the **Remember Status** command. An example of coordination axis creation is given in Section 3.3.13.15 on p. 1108.

A coordination axis can have additional designations (see Fig.3.3.122 on p. 1097). The sequence of their creation is described in Section 3.3.13.12.5 on p. 1102.

3.3.13.14. Coordination Circular Axis



To create coordination circular axis, call the **Coordination Circular Axis** command.

Specify a point of center of axis circle (**pC**). A ghost designation appears on the screen with the default parameters.

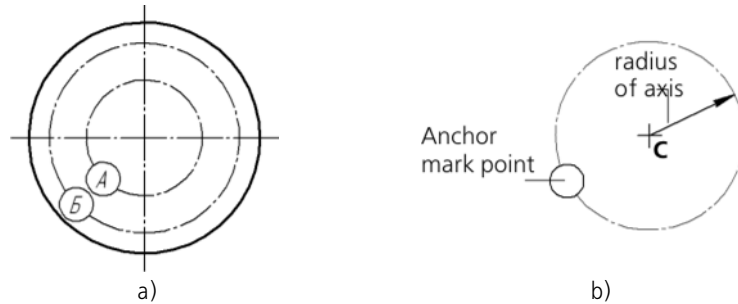


Fig. 3.3.129. Circular coordination axis:
a) usage example, b) axis parameters

Enter the symbol text.



To do automatic number Assignment while inputting the mark text, activate the **Auto Continue** switch (see Section 3.3.13.12.1 on p. 1097).

For details about coordination axis designation creation, see Section 3.3.13.12.2 on p. 1098.

Specify a point of allocation of the mark (**p**). The point **p** defines the value of radius of a circle. In the document the circle coordination axis with the radius equal to distance from center (**pC**) to a point of allocation of a brand (**p**) will be created. While creating a designation with the mouse, the radius is calculated automatically, and the value is input into the **Radius** or **Diameter** field.

Set up the coordinate circular axis parameters on the **Options** tab of the Property bar (see Section 3.3.13.14.1 on p. 1107).

You can create several designations during one call of the **Coordination Circular Axis** command.



While creating axes with the same parameters (all or a certain group), it is convenient to use the **Remember Status** command. An example of coordination axis creation is given in Section 3.3.13.15 on p. 1108.

A coordination axis can have additional designations (Fig. 3.3.130). The sequence of their creation is described in Section 3.3.13.12.5 on p. 1102.

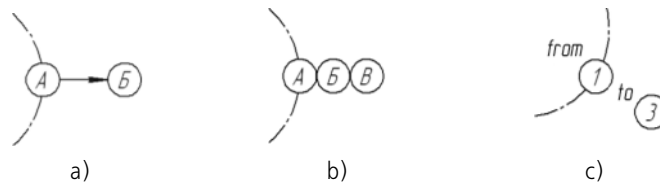


Fig. 3.3.130. The elements of additional designations for coordination circular axes:
a) orientation pointer, b) mark, c) text

3.3.13.14.1. Display Style Parameters

To set up the display style of a coordination circular axis, activate the **Parameters** tab on the Properties bar.



To disable marks on coordination axes, activate the **Mark** switch.

The default settings of the **Mark** switch are available in the **Coordinate Axes— Common Settings** item in the settings dialog of the current document (see Section 3.3.13.2 on p. 1069).

The settings of the rest of the control widgets are equal to those for straight axes. They are described in Table 3.3.11 on p. 1099.

3.3.13.15. Using the “Remember Status” Command to Create Axes



To construct several coordination axes of the same type, you can use the **Remember Status** command on the Advanced Control panel.



Automatic object creation should be embedded — the **Auto Create Objects** button on the Advanced Control panel must be pressed.

3.3.13.15.1. Straight Axes

To create straight axes, for example, long or transverse, do the following.

1. Call the **Straight Coordination Axis** command. Not specifying points **p1** and **p2**, make customizations on tabs **Parameters**, the **Hanging Indent 1**, the **Hanging Indent 2** of Panels of properties.
2. Set and fix values in fields the **Corner** and **Length** on a tab the **Coordination Axis**.
3. Click the **Remember Status** button.
4. Input the axis designation and activate the **Auto Continue** switch.
5. Specify consistently index points (**p1**) new coordination axes.

To create a grid of direct coordination axes (for example, in step 25 mm on axes OX and 37 mm on axis OY), fulfill following operations.

1. Go through steps 1-4 of the previous sequences of operations, having set value 0 in the field the **Angle**.
2. Specify an index point of the first horizontal axis the mouse or having entered co-ordinates in the field **p1**.
3. Without moving the mouse, enable the **p1** field and increase the current value of the Y coordinate by 37. For this purpose, press the following keys one by one (key combinations):

`<Alt>+<1>`, `<Tab>`, `<End>`, `<+>`, `<3>`, `<7>`, `<Enter>`.

The new axis displaced on 37 mm upwards concerning the first axis will be constructed. To construct the next axis, press (not shifting the mouse) the specified keys once again etc.

4. Set value 90 in the field the **Corner**, fix it and enter axis number.
5. Specify an index point of the first vertical axis the mouse or having entered co-ordinates in the field **p1**.
6. Not shifting the mouse, press consistently keys (shortcut key):

`<Alt>+<1>`, `<End>`, `<+>`, `<2>`, `<5>`, `<Enter>`.

To construct the next axis, press (not shifting the mouse) the specified keys once again etc.

3.3.13.15.2. Arc Axes

To create arc axes, for example, as arcs of concentric circles, do the following.

1. Call the **Coordination Arc Axis** command. Not specifying points **pC**, **p1** and **p2**, make customizations on tabs **Parameters**, the **Hanging Indent 1**, the **Hanging Indent 2** of Panels of properties.
2. Set and fix values in field, **Angle 1** and **Angle 2** on the **Coordination Axis** tab. Select the centerpoint.
3. Click the **Remember Status** button.
4. Input the axis designation and activate the **Auto Continue** switch.
5. Specify consistently the mouse of a point of the ends of radiuses of new coordination axes or consistently set values in the field **Radius**.

To construct two groups of concentric arc axes symmetric concerning a straight line (for example, axes OY), fulfil following operations.

1. Build the first axis group by performing steps 1–5 of the operations sequence described above.
2. Edit value in the field **Angle 1**, having transformed it in a difference between 180 and current value. For this purpose, enable the field and, not changing current value, enter before it *180* characters and press the *<Enter>* key.
3. In the same way edit value in the field **Angle 2**.
4. Replace a direction of construction of an axis.
5. Make active the field **Center** click under its name and specify centre of new group of axes.
6. Specify consistently the mouse of a point of the ends of radiuses of axes or consistently set values in the field **Radius**.



To place the second group symmetrically relative to the OX axis (step 2), change the signs of the numbers in the **Angle 1** and **Angle 2** fields.

3.3.13.15.3. Circular Axes

To create circular axes, for example, as concentric circles, do the following.

1. Call the **Coordination Circular Axis** command. Not specifying points **pC** and **p**, make customizations on a tab **Parameters** of the Panel of properties.
2. Specify the mouse a point **pC** or set and fix centre co-ordinates in input fields **Center** on a tab the **Coordination axis**.
3. Click the **Remember Status** button.
4. Input the axis designation and activate the **Auto Continue** switch.
5. Specify consistently points of layout of marks (**p**) new coordination axes. To precisely position the cursor, use snaps.

3.3.14. Automatic Creation of Views and Hyperlinks

3.3.14.1. Automatic Creation of View

Once some designations are created, the view creation command is run immediately. ¹View property controls appear in the Properties bar. If necessary, edit the view parameters and specify the view's base point. A view will appear in the document; its caption will be associatively related to the designation of the layout object you have just created (for more details, see Section 3.5.4.6.2 on p. 1166).

This option is available when creating the following designations:

- ▼ View Arrow,
- ▼ Cut/Section Line,
- ▼ Enlarged Detail,
- ▼ Sectional Node,
- ▼ Node.

If the designation has been created in a non-associative view, the new view will also be non-associative. Once designation of the enlarged detail or node is created, the image of the enlarged detail or node is automatically created in the view. In other cases, the new non-associative view will be empty.

If the designation has been created in the associative view, the new view will also be associative:

- ▼ for a view arrow- **View in Arrow Direction**,
- ▼ for an enlarged detail- **Enlarged Detail**,
- ▼ for a cut/section line- **Cut/Section**,
- ▼ for a node- **Enlarged Detail**.

The result associative view contains the automatically generated model image that corresponds to the designation, i.e. view in arrow direction, cut/section, or enlarged detail.



Creation of a sectional node's designation results in a non-associative view.

For more information about views, see Section 3.5.1.2 on p. 1147 (non-associative views) and 3.6 on p. 1205 (associative views).

The automatically created image of an enlarged detail or node has a number of peculiarities (see Section 3.3.14.1.1 on p. 1111).

-
1. After mechanical engineering designations are created, the view is generated unconditionally; after civil engineering designations are created, the view is generated only if the **Create View** option of the Property Bar was enabled during their customization.

3.3.14.1.1. Automatically created image of an enlarged detail or node

The automatically created image of the enlarged detail or node contains copies of objects contained inside the designation contour. The following objects are copied:

- ▼ geometrical objects,
- ▼ drawings, fragments and images insertions from views of other drawings,
- ▼ designations:
 - ▼ center mark,
 - ▼ wavy line,
 - ▼ line with bends,
 - ▼ conditional intersection,
 - ▼ auto centerline,
 - ▼ centerline by two points.

Other designations and dimensions are not copied. Copied objects cannot be edited or deleted but it is possible to move them to a different layer and change their style.

The style of the enlarged detail's contour can also be modified. You can also delete the contour. In order to restore the contour, call the **Restore Border** command from the context menu of the enlarged detail selected in the Drawing Tree or in the document's window.



If a view containing the image of the enlarged detail or node is non-associative, the changes in the source view are not traced, while if the view is the associative one, only those changes that result from model editing are traced.

3.3.14.2. Automated Creation of Hyperlink

When creating a new designation of

- ▼ a view arrow,
- ▼ an enlarged detail,
- ▼ cut/section lines,
- ▼ a node,
- ▼ a sectional node,

a hyperlink may be created automatically to link the designation with the auto-created view.

To create an automated hyperlink, the following conditions must be met:

- ▼ when placing a designation, a view linked with this designation should have been automatically created (see Section 3.3.14.1),
- ▼ the option **Automatic hyperlinks creation** should have been enabled in the hyperlink setup dialog (see Section 9.1.2.7 on p. 1675).



If at least one of the conditions is not met, the hyperlink is not created automatically. If necessary, you can create a hyperlink by selecting an object and calling the **Hyperlink...** command... from the **Insert** menu or from the context menu (see Section 11.2.2.1 on p. 2018).

3.4. Edit

3.4.1. General Techniques of Editing

KOMPAS-3D offers various option for the user to edit objects.

- ▼ The simplest and most used techniques of editing — moving and copying objects, handles movement — can be performed using the mouse. Editing methods are described in Section 3.4.1.1 on p. 1114.
- ▼ Changing and Copying Object Properties can be managed by **Properties** window. Methods on operation with object properties are described in Section 3.4.1.2 on p. 1117.
- ▼ Operations with objects, such as deleting an object's part, transformation of objects, copying along grid and so on, are performed using special commands.

The commands of editing geometric objects are grouped in the **Editor** menu, while the command buttons are located in the **Edit bar** (Fig. 3.4.1).

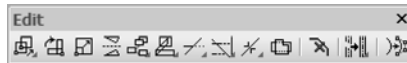


Fig. 3.4.1. Edit **bar**

Before calling the commands of moving, rotating, scaling, symmetry transformation and copying, you must select the objects participating in the operation (the selection of objects is described in Section 3.1.3 on p. 876). Editing commands work is described in Sections 3.4.2 – 3.4.8.

Besides that, you can edit object parameters. For this purpose, double-click an object or press *<Enter>* or *<Space>* after highlighting the object (the cursor should not be positioned over the object). The Property bar displays the same set of control elements as the set available when creating an object. You can edit object properties: change any of its properties and characteristics.

Sometimes you may need to edit only the text included in an object, i.e., a dimension text, text in braces, surface finish symbols, etc. In these cases, it is convenient to activate the Text Entry dialog without activating the process of object editing. This can be completed by the following methods:

- ▼ double-click the text (the text itself, not the lines or points making up the object);
- ▼ select the object, move the cursor so that it is located above the object and press *<Shift> + <Enter>* or *<Shift> + <Space>*.



On the **Edit bar**, buttons for calling the **Delete multiline's line break** and **Delete all breaks in the specified multiline's line** commands are located. More information about working with them is provided in Section 3.2.12.11 on p. 954.

Objects can be copied together with attributes and properties and without them. Detailed description of copying objects with attributes can be found in Section 12.3.3.3 on p. 2068. Details of copying objects with predefined user properties can be found in Section 5.1.4.3 on p. 1394.

Such object properties as layer, current line style, text style, etc., can be copied between objects, i.e. move from one object to another (see Section 3.4.1.2.2 on p. 1119).

3.4.1.1. Editing Objects with a Mouse

3.4.1.1.1. Moving Objects with a Mouse

To move objects with a mouse, perform the following operations.

1. Select the objects to move. Selection of geometric objects is described in Section 3.1.3 on p. 876, selection of views is described in Section 3.5.5.4 on p. 1173.
2. Place the cursor on one of the selected objects (but not on a handle) and click the left mouse button.
3. Holding the mouse button, drag the objects. Object ghost following the cursor will be displayed on the screen.
4. Once the desired object position is reached, release the mouse button.

The objects will be moved from their old locations to the new ones.

3.4.1.1.2. Copy Object with a Mouse

To copy the objects with a mouse, perform the following operations.

1. Select the objects to copy.
2. Press the *<Ctrl>* button.
3. Holding the *<Ctrl>* key, place the cursor on any of the selected objects (but not on a handle), press the left mouse button and move the mouse.
4. Release the *<Ctrl>* button and the mouse button.

Object ghost following the cursor will be displayed on the screen.

5. Move the mouse until the required location of the objects is reached, then click the left button. The objects will be copied to the specified location, while the originals will stay at their old place.

You can continue to copy objects fixing the location of each next copy.

6. To finish copying, press the *<Esc>* button.

3.4.1.1.3. Editing Defining Points

You can change the configuration of a geometric object or layout object by moving either of its handles.

To go into the mode of editing the object, click that object with the mouse.

In this mode, the characteristic points are displayed in the form of small black squares, and the object is highlighted.

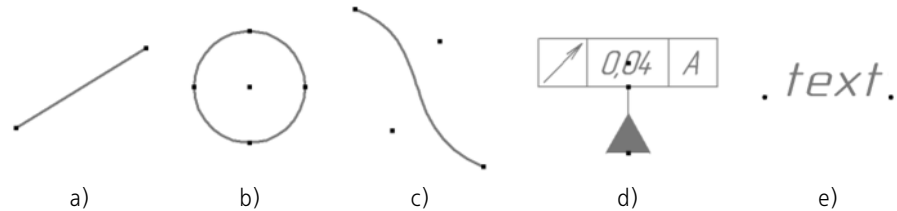


Fig. 3.4.2. Defining points of:
a) segment, b) circle, c) spline, d) geometric tolerance, e) text

Arc's handles also have black triangles. If you move the triangles, only the arc's radius will be modified.



Fig. 3.4.3. Arc's handles

When you move cursor to the handle, the cursor assumes the following shape:



— if the handle can be moved as needed,



— if movement of the handle is limited or if it can have only several fixed positions,



— if movement of the handle results in rotation of the object or some part of it.

If several objects are selected and their handles coincide, they will be displayed as a common handle. If the common handle is moved, the respective handles of each object will also be moved. Setup of common handles' display and the number of selected objects are described in Section 9.1.7.12.

The following methods can be applied to move the handle:

- ▼ **"dragging" the selected object's handle with the mouse,**
- ▼ **activation of the handle and specifying its new position.**

Methods of moving handles are described below.

To clear the highlighting of the object after moving its handles, click the mouse outside this object's image. When an object is deselected, its defining points also disappear.

"Dragging" the selected object's handle with the mouse

In order to "drag" a handle, hover the cursor over it, left-click it and move the mouse without releasing the mouse key. The handle will move after the cursor. When the required position is reached, release the mouse button.

If rounding is enabled (see Section 1.4.2.14), moving the object's handle will result in increment changing of its linear and angular parameters. The increment will correspond to the cursor's step value. Current values of parameters are displayed beside the cursor. Text's background beside the cursor can be adjusted (see Section 9.1.3.3).

If the mouse is not moved during a second after positioning the cursor on a handle and clicking the mouse button, auxiliary paths will be displayed. When moving a handle along the auxiliary path, only one object's parameter will be modified (Fig. 3.4.4).

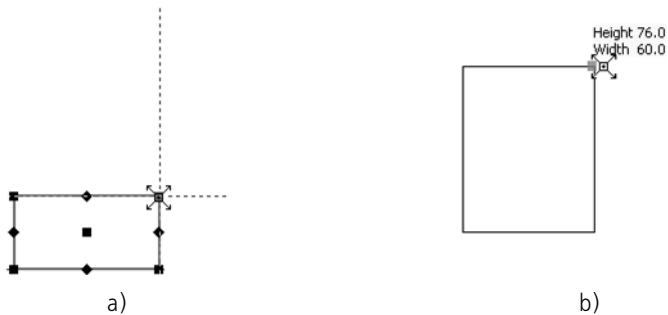


Fig. 3.4.4. Auxiliary paths: a) starting position of the handle, b) “dragging” the handle along the path

If, when “dragging” a handle, the cursor is moving from the auxiliary path, the path will disappear from the screen.

If you want auxiliary paths to be displayed permanently when “dragging” handles, press and hold the `<Ctrl>` key.

In order to disable displaying auxiliary paths when “dragging” handles, press and hold the `<Alt>` key.

Auxiliary paths are displayed during editing of:

- ▼ a line segment,
- ▼ multilines,
- ▼ arcs,
- ▼ an ellipse, ellipse's arc,
- ▼ a rectangle.

Activation of a handle and specifying its new position

You can activate a handle by clicking the left mouse button or by pressing the `<Enter>` key.

After activation the point moving process is started. Activated handle will be highlighted, and the zero value of offset parameters will be indicated near the cursor. The system will expect specification of the position of the point which is “linked” to the cursor. During the cursor's movement, changing coordinates are displayed next to it, as well as distance and angle with respect to the point's original position.

Select the new point position using one of the methods:

- ▼ click the mouse in the required place of document,
- ▼ set up offset parameters values on the Properties bar,

- ▼ move the point using the mouse the arrow keys and fix the point,
- ▼ enter the cursor coordinates on the **Current Status** toolbar (see Section 1.4.1.1 on p. 75) and fix the point.

You can fix the point by clicking the mouse or pressing `<Enter>`.

You can limit the point's movement by fixing a required parameter on the Property bar. For example, in order to move a point in the direction of the X axis, set zero value in the Y coordinate's field and fix it. Ghost of the straight line, along which the point can be moved, will be displayed on the screen.

When selecting a point position you can use the Snaps (see Section 3.1.1 on p. 865) and the Geometrical Calculator (see Section 3.1.2 on p. 871).

Moving process will be automatically finished after you fix a point.



To interrupt the process without shift, click the **Interrupt Command** button on the Advanced Control panel or the `<Esc>` button.

3.4.1.2. Changing and Copying Object Properties

You can view and change the properties of the selected objects without editing each separate object, and you can copy the properties among the objects.

You can change and copy the following object properties:

- ▼ The layer on which the object is located
- ▼ some parameters, which are set on the Property bar when creating or editing this object,
- ▼ the text included into the object contents,
- ▼ the text style, font and paragraph parameters,
- ▼ the spelling language attribute,
- ▼ hyperlinks.

To change the properties of the selected objects, as well as to review the properties of a source and select the properties to be copied, use the **Properties** window. The techniques of property changing in this window differ from those of property copying.

The **Properties** window and general procedure for modifying the properties are described in Section 1.4.2.2 on p. 82. Peculiarities of working in the **Properties** window for graphical documents are considered in Section 3.4.1.2.1, copying properties between objects is described in Section 3.4.1.2.2.

3.4.1.2.1. The Properties window in graphical documents

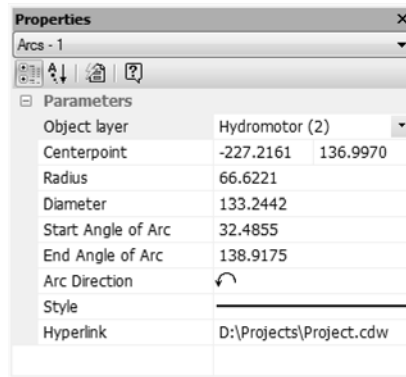


Fig. 3.4.5. Arc's properties



The **Properties** window is displayed by calling the **Editor – Properties** command or by clicking the **Properties** button on the Standard bar.



It is convenient to call the **Properties** command from the context menu of the selected objects.

Controls of the **Properties** window and modifying the properties are described in Section 1.4.2.2 on p. 82.

Working with graphical documents has the following specifics.



- ▼ In order to specify objects, the **Select by properties** button in the **Properties** window's toolbar can be used. It serves as a command button for selecting objects by a set of properties, excluding objects from the selected ones by a set of properties, and inverting the selection (for details, see Section 3.1.3.3 on p. 879).
- ▼ Within a drawing, objects can be specified in the Drawing's Tree:
 - ▼ graphical macroelements, fragments and images insertions from views of other drawings;
 - ▼ models by which the drawing is created.
- ▼ When changing properties, their specifics and type of selected objects should be taken into account.
 - ▼ The property settings for length (line length, circle radius, etc.) and for point coordinates are displayed in the current units of measurement.
 - ▼ Properties which changing makes no sense at the moment are unavailable. For instance, there is selected hatch with the **Area** filling type. The **Band Width** and **Arranging Hatch with Respect to Contour** properties are shown on the property sheet but their values cannot be changed. These properties become available after you change value of the **Filling Type** property from **Area** to **Band**.

- ▼ Changing the **Designation** property is not possible for view arrows, detail drawings, cross-section axes and bases, if auto-sorting of lettered items in the designation was enabled during their creation or editing (see Section 3.3.12.20 on p. 1067).
- ▼ The **Object Layer** property and properties being coordinate and angle values are available if selected objects belong to one view.
- ▼ The values of coordinates and angles are measured in a coordinate system of a view containing selected objects or in its current local coordinate system (if any)— even if the view, to which the selected objects belong, is not current at the moment.
- ▼ The **Language** property is available if a text or a table is highlighted.
- ▼ The **Hyperlink** property is available if one object is highlighted, or if all highlighted objects contain similar hyperlinks, or neither of selected objects contains the hyperlink.
- ▼ You can copy properties between objects, including model's objects in a drawing. Copying methods are described in Section 3.4.1.2.2 on p. 1119.

3.4.1.2.2. Copying Properties

You can copy the properties of a specified object (source of properties) to other objects, i.e., make the properties of other objects the same as those of the specified object.

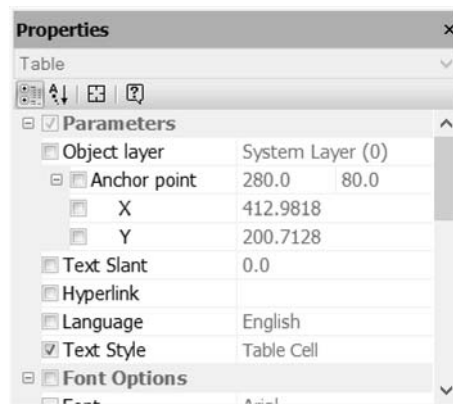


Fig. 3.4.6. Selecting properties for copying in the **Properties** window

The properties are copied according to the following rules:

- ▼ The **Object Layer** property and the properties, which are the values of coordinates and angles, are copied in a drawing among the objects belonging to the same view.
- ▼ The **Style** property is copied among all objects allowing the change of a line style (for example, a circle line style can be copied into a line, equidistant line, wavy line, a brace, etc.).
- ▼ properties from **Text** (or **Text Style**), **Font Options** and **Paragraph Properties** groups are copied among all objects that contain text and allow changing of these properties,
- ▼ The **Language** property is copied among texts and tables,
- ▼ The **Hyperlink** property is copied among all objects in which the hyperlink may be inserted.

- ▼ Other properties are copied only between objects of the same type (e.g. leader arrow view can be copied to another leader only and not to a dimension).

By default, copying is disabled for some object properties. Those include **Object Layer** property, values of defining points coordinates and other properties, copying of which is usually not required. Enabling (and disabling) of copying properties is done in the **Properties** window. If it is not displayed on the screen, enable it by clicking the **Properties** button.



To copy the properties, perform the following operations.



1. Apply the **Edit— Copy Properties** command and press the **Copy Properties** button on the Standard panel.

2. Specify a property source object. To do it, select it in the document window. The source object may be specified both before and after you call the **Copy Properties** command.

The name of the source object and the current values of its properties will be displayed in the **Properties** window. The control options for copying properties are located to the left of the properties' names. When you enable an option, the property corresponding to it will be copied from the source object to the specified object(s), and when you disable it — the property will not be copied. (Fig. 3.4.6).

3. Specify the object(s) to copy source properties to. To do it, click on the objects or select them by a frame.

Properties of objects will be changed.

A source object remains selected during copying its properties.



To change the source of properties, press the **Select Again** button on the Advanced Control panel of the **Properties** window and then specify a new source object.



4. To finish copying properties, click the **Interrupt Command** button on the Advanced Control panel or the <Esc> button.

You can copy object properties among different graphic documents which are open in one KOMPAS-3D window. You can do it in the same way as copying properties among the objects of one document. The only condition for performing this operation: no other command shall be activated in the document into which the properties are copied.

3.4.2. Shift

3.4.2.1. Arbitrary Move



To move the selected objects, call the **Move** command.

- ▼ If the post-move location of any point of the image is known, set it up as a base point (**p1**). Then set a new location for this point— **p2**.
- ▼ If the moves of the objects along the axes of the current coordinate system are known, enter them in the corresponding fields on the Property bar.

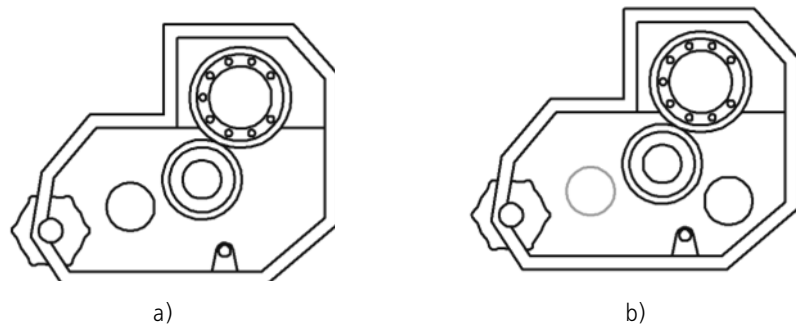


Fig. 3.4.7. Arbitrary move: a) source image, b) result of the operation

In a number of cases, it is more convenient to simply drag the selected objects with the mouse without calling the move command. This option is detailed in Section 3.4.1.1.1 on p. 1114.



Sometimes it is required to move only some geometric objects forming an image, and the rest— to modify in a corresponding manner. Such editing is performed by use of the **Transformation by Move** command (see Section 3.4.5.2 on p. 1134).

3.4.2.1.1. Controlling the Source Objects

Upon completing the operation, its source objects may be left in the document or deleted.

The source objects are controlled by use of the group of switches **Mode** on the Property bar.

To ensure that the source objects will be automatically deleted upon completion of the operation, activate the switch **Delete Source Objects**. The activation of the switch **Retain Source Objects** means that they will be saved.



3.4.2.2. Move by Angle and Distance



To move the selected objects by a certain distance in a specified direction, call the **Move by Angle and Distance** command.

In the respective fields on the Property bar, enter the distance of the move and the angle between the radius-vector determining the direction of the move and the abscissa of the current coordinate system.

A ghost of the moved objects will appear on the screen.

The values of the shifts along the axes of the current coordinate system will be calculated automatically and displayed in the respective info fields on the Property bar.

The source objects are controlled in the same way as in the case of arbitrary move.

In order to fix the ghost, press the **Create Object** button.



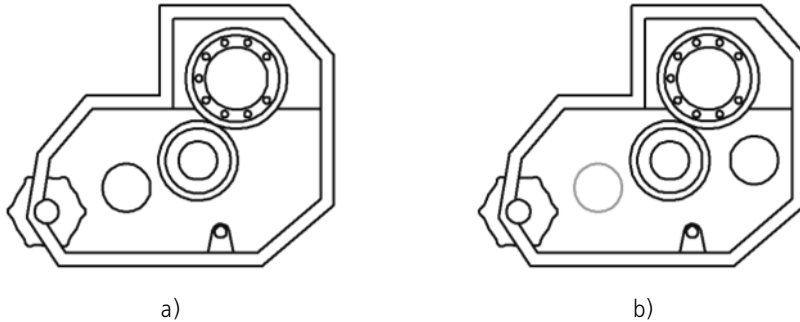


Fig. 3.4.8. Move by angle and distance: a) initial image, b) move of a hole by 22 mm at an angle of 10° to the X axis

3.4.3. Copying

3.4.3.1. Arbitrary Copy



To copy the selected objects, call the **Copy by Selection** command.

Specify the anchor point **p1** for copying.

Select the **p2** point defining the new anchor point position.

The shift values along the axes of the current coordinate system will be automatically calculated and displayed in the **Offset on X** and **Offset on Y** fields.

The control of the source objects is described in Section 3.4.2.1.1 on p. 1121.

Upon fixing the new location of a base point, the system copies the selected elements to a specified place and waits for specifying the next place for copying.

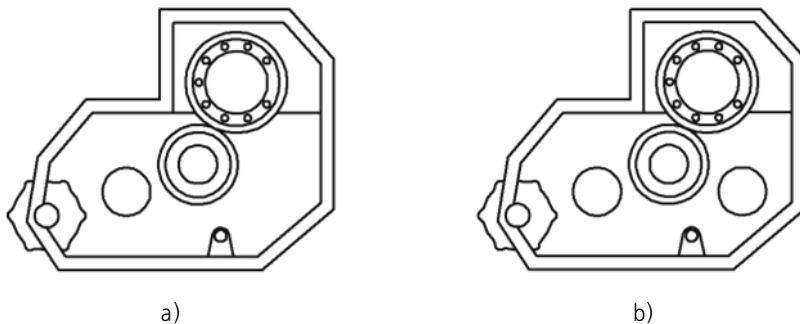


Fig. 3.4.9. Arbitrary copying: a) source image, b) result of the operation



Sometimes it is more convenient to perform simple copying of the selected objects with a mouse, without the use of a special command. This option is detailed in Section 3.4.1.1.2 on p. 1114.

3.4.3.1.1. Scale and Rotation of Copies

By default, the object copies have the same dimensions and orientation as those of the source object (Fig.3.4.10, a).

You can scale and/or rotate, where applicable, the copies relative to the source objects (Fig.3.4.10, b). To do it, enter the required values in the fields **Angle** and **Scale** on the **Copy** tab of the Property bar.

When copying and scaling, you can specify whether to scale the extension lines and the dimension leaders (if those are present among the copied objects). The procedure of the extension lines scaling is described in Section3.4.4.2.1 on p. 1131.

When copying is performed along a curve, parallelogram and circular patterns, the scaling and rotation of the copies are performed in a similar way.



Fig. 3.4.10. Copying objects: a) maintaining the rotation angle and the scale, b) by changing the rotation angle and the scale

3.4.3.2. Copying along Curve



To create a pattern of copies of the selected objects locating them along a specified curve, call the **Copying along Curve** command.

The control of the source objects is described in Section3.4.2.1.1 on p. 1121.

Specify the **p1** anchor point for copying.

With the cursor, specify the curve, along which to copy the objects.

Enter the number of copies and their step in the respective fields on the Property bar.



The step is measured on the curve along which the copying is performed.

Specify the required step interpretation, the location of the copies relative to the normal to the curve and the direction of copying. These parameters are described in detail in the next sections.

On the curve, select a starting point for copying, it will coincide with the base point of the first instance of the pattern.



The screen will display a ghost of the set of copies. To fix that, press the **Create Object** button.

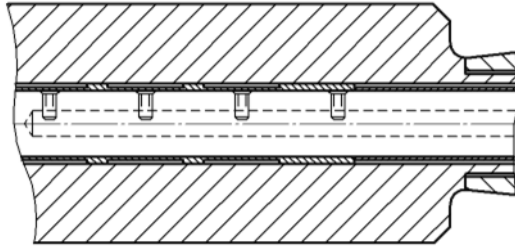


Fig. 3.4.11. Example of tracing along a curve

3.4.3.2.1. Step Interpretation



By default, the value entered in the **Step** field is treated as the distance between the respective points of the neighboring pattern instances. In this case, in the group **Mode**, the switch **Distance between Adjacent Copies** is active. Another switch in this group is **Distance between End Copies**. Activate it, if you need to uniformly place copies, the number of which is specified in the **Number** field, on the curved line section, the length of which is specified in the **Step** field.

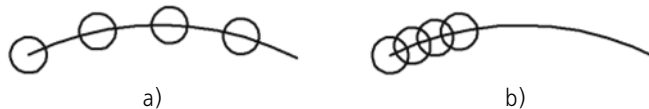


Fig. 3.4.12. Copying circles along an arc:
 a) **Distance between Adjacent Copies** switch is active,
 b) **Distance between End Copies** switch is active

3.4.3.2.2. Allocation of Tracings



By default, the copies are additionally rotated to the normal to the curve: each copy is rotated in such a way that its location relative to the normal to the curve drawn in the insertion point would coincide with the location of the source object relative to the Y axis of the absolute coordinate system. In this case, in the **Normal** group, the switch **Follow Path** is active. If all copies must be located in the same way as the source object, activate the **Do not turn in up to the Normal** switch.



Fig. 3.4.13. Copying an ellipse along a spline: a) without additional rotation of the copies to the normal,
 b) with additional rotation of the copies to the normal



The pattern instances are arranged in the already described manner if the **Angle** has zero value (see Section 3.4.3.1.1 on p. 1123). Otherwise, each copy will be additionally rotated by the specified angle.

3.4.3.2.3. Tracing Direction



By default, the objects are copied along a curve in counterclockwise direction from the starting point. In this case, in the **Direction** group, the **Negative Sense** switch is active. To have the array placed in the different direction from the start point, activate the **Positive Direction** switch.



Obviously, the selection of the array creation direction makes sense only in the cases when the starting point is not an end point of the curve.

3.4.3.3. Copy along Parallelogram Grid



In order to create a pattern of copies of the selected objects locating them in grid nodes with specified settings, apply the **Copy along Grid** command.

The control of the source objects is described in Section 3.4.2.1.1 on p. 1121.

Specify the base point of the copied objects **p1**. In the process of pattern formation, the copies will be placed in a way that their base points coincide with the grid nodes (see Fig. 3.4.14, 3.4.15).





The pattern ghost will be displayed. To set the grid up as required, activate the **Options** tab of the Property bar. The controls located on it are presented in Table 3.4.1. The pattern parameters, which control the above-mentioned controls, are shown on Fig. 3.4.14.

Tab. 3.4.1. Control elements for parallelogram grid

Element	Description
Slope	A slope of the first axis of the grid to the abscissa of the current coordinate system.
N1 N2	Number of pattern instances along the first and second grid axes.
Aperture angle	Angle between grid axes.
Step 1 Step 2	Step of copies along the first and second pattern axes.
Mode 1 Mode 2	Switches controlling the step interpretation along the axes. The step interpretation procedures are described in Section 3.4.3.2.1 on p. 1124.



Tab. 3.4.1. Control elements for parallelogram grid

Element	Description
 	Instances in Grid Nodes Switches that control the way of placement of the copies. These groups of switches are accessible if the number of copies along each of the axes is greater than or equal to three.
 	Instances Inside Grid Switches that control the way of placement of the copies. These groups of switches are accessible if the number of copies along each of the axes is greater than or equal to three.

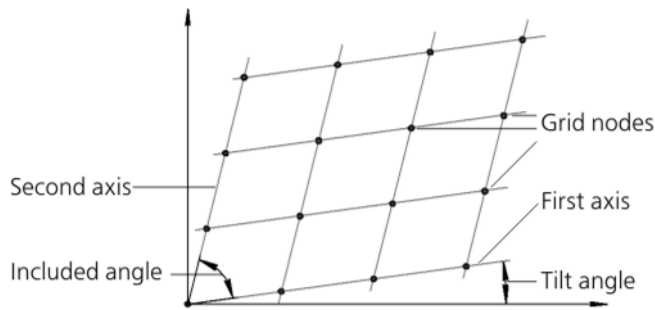


Fig. 3.4.14. Formation scheme of parallelogram grid

Any change to a pattern or grid parameter is immediately reflected on its ghost.

To fix a ghost, select the point of pattern insertion **p2**.

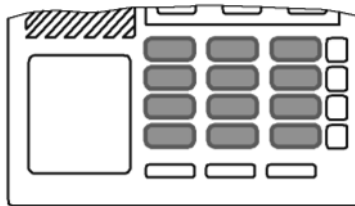


Fig. 3.4.15. Example of copying along grid

3.4.3.4. Copy along a Circular Grid



To create a pattern of the selected objects locating them in nodes of a circular grid, call the **Copy along Circular Grid** command.

The control of the source objects is described in Section 3.4.2.1.1 on p. 1121.

Specify the base point of the copied objects **p1**. In the process of pattern formation, the copies will be placed in a way that their base points coincide with the grid nodes (see Fig.3.4.17, 3.4.18).

The pattern ghost will be displayed. To set the grid up as required, activate the **Options** tab of the Property bar. The controls located on it are presented in Table 3.4.2. The pattern parameters, which control the above-mentioned controls, are shown on Fig.3.4.16.

Tab. 3.4.2. Control elements for a circular grid

Element	Description
Radius	The value of the radius of the source grid circle.
Step 1 Step 2	Step of the copies along the radial and circular directions.
Mode 1 Mode 2	Switches controlling the interpretation of the step along the radial and circular directions. The step interpretation procedures are described in Section 3.4.3.2.1 on p. 1124.
N1 N2	Number of the copies along the radial and circular directions.
Starting Angle	The angle between the abscissa of the current coordinate system and the first radial line of the grid.
Copy in Center	A group of switches controlling the drawing of the copies in the grid center point. If the switch Retain Copy in Center is active, one more instance will be added to a pattern so that its base point would coincide with the grid center. By default, the formation of a central copy is deactivated.
Copies Orientation	A group of switches controlling the location of the copies. Its functioning is described in details below.

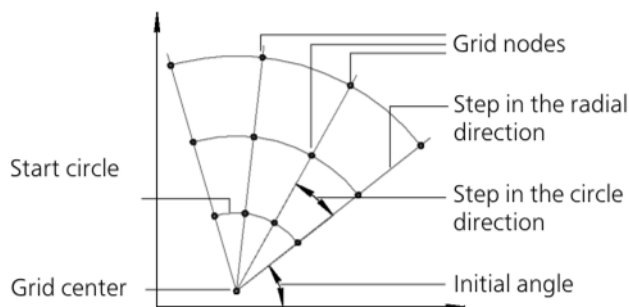


Fig. 3.4.16. Procedure to create a circular grid

Any change to a pattern or grid parameter is immediately reflected on its ghost.
To fix a ghost, select the point of pattern insertion **p2**.

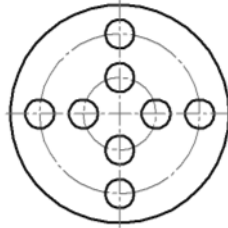


Fig. 3.4.17. Example of copying along circular grid

3.4.3.4.1. Allocation of Tracings



By default, in the group **Copies Orientation**, the switch **Set Copies Radially** is active. In this case, each copy is rotated around its base point so that its position relative to the radial line, on which it is located, would coincide with the location of the source object relative to the X axis of the global coordinate system. If all copies must be located in the same manner as the source object, activate the **Do Not Set Copies Radially** switch.

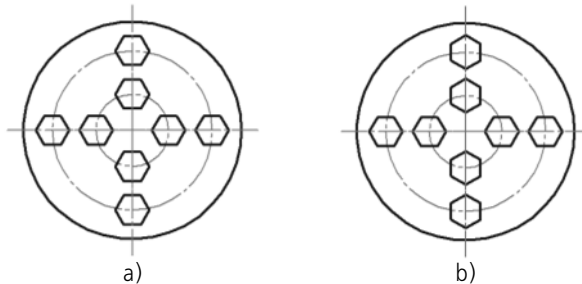


Fig. 3.4.18. Copying along a circular grid:

a) without additional turning to the radial direction, b) with additional turning to the radial direction



The pattern instances are arranged in the already described manner if the **Angle** has zero value (see Section 3.4.3.1.1 on p. 1123). Otherwise, each copy will be additionally rotated by the specified angle.

3.4.3.5. Copy along Circle



To create a pattern of copies of the selected objects locating them on the circumference with a specified center, call the **Copy along Circle** command.

Set a center point for copying.

A ghost array appears on the screen with the default parameters.

Enter the total number of pattern instances in the respective field on the Property bar.



Since the source object is included in the circular pattern composition, then the number of the newly created copies will be one less than the entered value.



A group of switches **Mode** allows the selection of a method of placement of the pattern instances (Fig.3.4.19).

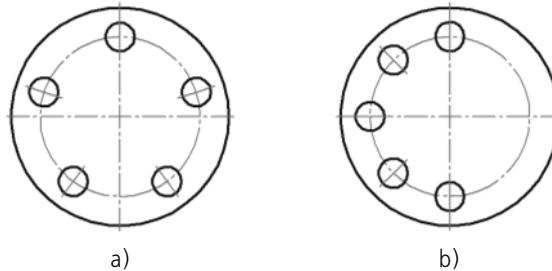


Fig. 3.4.19. Copying mode: a) over the entire circle, b) with angular step of 45°



If you select the location of copies with a specified angular step, enter its value in the field **Step** and set the required direction of the copying using a group of switches **Direction**.



Any change to an array parameter is immediately reflected on its ghost.



To fix a pattern, press the **Create Object** button.

3.4.4. Object Transformations

3.4.4.1. Rotation



To rotate the selected objects, apply the **Rotation** command.

Select the center point of rotation.

- ▼ If the post-move location of any point of the image is known, set it up as a base point (**p1**). Then set a new location for this point— **p2**.
- ▼ If the rotation angle of the objects is known, enter it in the corresponding field of the Property bar.

The control of the source objects is described in Section3.4.2.1.1 on p. 1121.

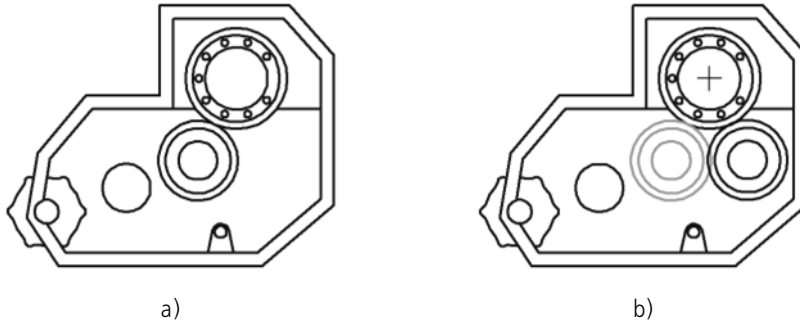


Fig. 3.4.20. Performing rotation: a) initial image, b) resulting image after rotation of a hole (the center of rotation is marked with a cross)



Sometimes you need to rotate only some geometric objects forming an image, and to modify, as appropriate, the rest of them. Such editing is performed by use of the **Transformation by Rotation** command (see Section 3.4.5.3 on p. 1134).

3.4.4.2. Scaling



In order to perform scaling of the selected objects, apply the **Scaling** command.

Enter in the corresponding fields of the panel the values of the scaling coefficients in the direction of the coordinate axes (you can enter different values of the scaling coefficients along the axes).



You cannot enter the Y-axis scale if there are circles, or arcs, or whole views among the selected objects. In this case, the operation is executed with the scale value along the Y-axis equal to the scale along the X-axis.

The control of the source objects is described in Section 3.4.2.1.1 on p. 1121.

A group of switches **Extension Lines** controls the scaling of the extension lines. It is detailed in the next section.

Select the center point of the scaling.

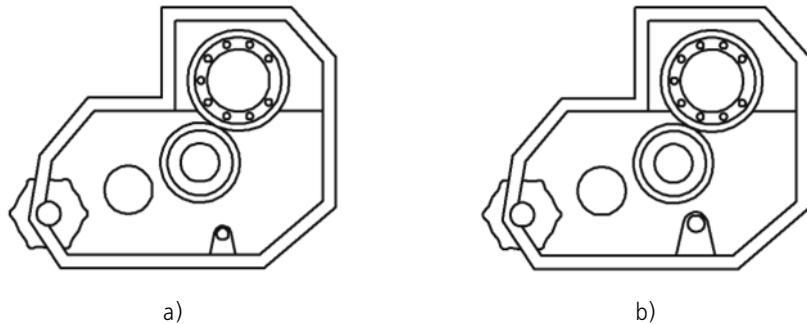


Fig. 3.4.21. Execution of the scaling: a) initial image, b) image after scaling an eyelet



Sometimes you need to scale only some geometric objects forming an image and modify properly the rest of them. Such editing is performed using the **Transformation by Scaling** command (see Section 3.4.5.4 on p. 1135).

3.4.4.2.1. Controlling the Scaling of the Extension Lines



By default, the dimension extension lines and leaders (if they are available among the objects participating in an operation) are not scaled, i.e., their length remains the same as in the source image. In this case, a switch **Do Not Scale** is active in the group **Extension Lines**.



If it is required to change the length of the extension lines and leaders in accordance with the preset scaling factors, activate the **Scale** option.

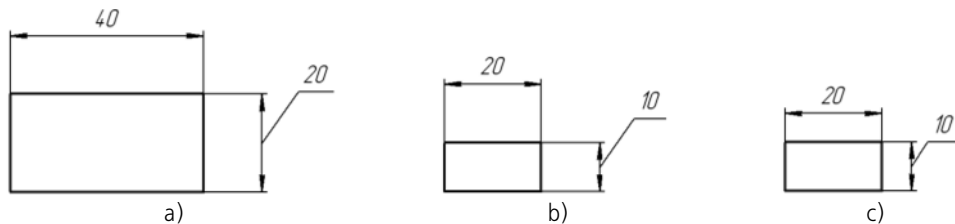


Fig. 3.4.22. Scale of the copies is reduced by a factor of two: a) source object, b) copying without scaling the extension lines, c) copying with scaling the extension lines

We recommend using scaling of the extension lines when the scale of the objects changes considerably and there are many dimensions among the objects.

3.4.4.3. Symmetry



To perform symmetry transformation relative to a line for selected objects, call the **Symmetry** command.

Set the first point belonging to the axis of symmetry (**p1**).

- ▼ If the location of the second point on the line (**p2**) is known, set it.

- ▼ If the slope of the axis (the angle between this axis and the abscissa of the current coordinate system) is known, enter it in the respective field on the Property bar.

The control of the source objects is described in Section 3.4.2.1.1 on p. 1121.

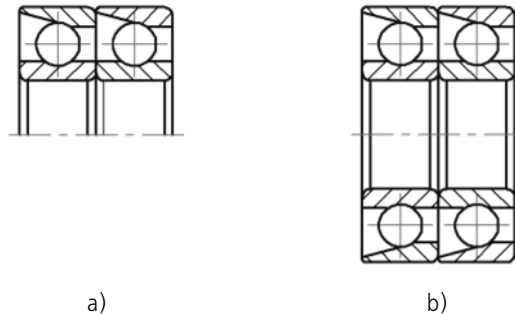


Fig. 3.4.23. Symmetry transformation: a) initial image, b) result of the command

3.4.4.3.1. Specifying the Existing Symmetry Axis

If a document already contains a straight line object serving as symmetry axis for the transformation to be performed, you can select this very object, not the points belonging to it. To do it, press the **Select Reference Object** button and select the required object with the cursor.



3.4.4.4. Conversion to NURBS

Conversion to NURBS (non-uniform rational B-spline) is possible for any geometric object or text. Such a conversion may be required for the editing of the next flexible object by moving its defining points.



To convert a geometric object or a text to NURBS, call the **Convert to NURBS** command. Select the object to be converted.

The control of the source objects is described in Section 3.4.2.1.1 on p. 1121.

The result of converting text into NURBS curves depends on the type of the font used. For example, when processing TrueType fonts, outlines are created for the letters, while when processing vector fonts, sets of line segments that form the letters are generated (Fig. 3.4.24).

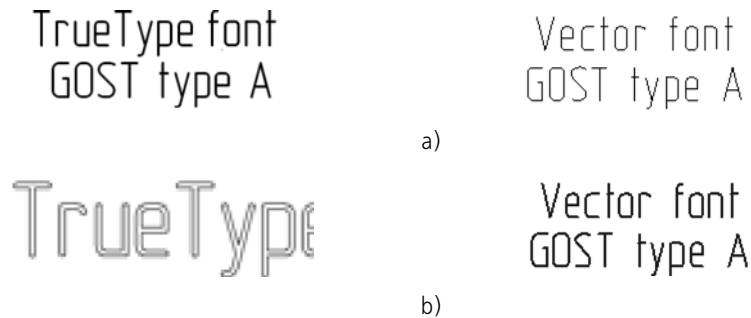


Fig. 3.4.24. Conversion of texts to NURBS: a) source objects— texts, typed with a TrueType font and vector font, b) results of the conversion



Some objects cannot be converted into a single NURBS without multiple points. The result of converting such objects is an outline consisting of several NURBS without multiple points.

3.4.5. Deformation

The deformation commands are used when it is required to move, rotate or scale a part of an image so that the objects, the defining points of which have changed their location, would not lose the link with fixed objects. This means that the deformation commands will make it possible to edit the elements without “breaking” an image.

After applying a deformation command, the system waits for specifying the objects to be transformed (i.e., you do not need to select the objects in advance, for example, as in the case of copying). After all objects are selected, the mode of performing the deformation command is activated. In this case, the set of control elements on the Property bar changes.

The procedure of selecting objects is the same for all three commands. This procedure is described in Section 3.4.5.1. Other sections describe the sequence of the operations to be performed for a deformation of a specific type.

3.4.5.1. Selecting Objects for Deformation

The selection of the objects to be deformed is done by using a rectangular marquee frame.

Select the opposite vertices of the marquee frame **p1** and **p2**. The “captured” objects will be selected. The system enters in Deformation mode.

You can change the set of selected objects using the buttons on the Advanced Control panel.



The button **Select by New Marquee Frame** serves to make a marquee frame. After clicking it, the previous highlighting is canceled, and the system is waiting again for object selection: the **p1** and **p2** fields appear on the Property bar. You can select the vertices of the marquee frame again.



The **Exclude/Add Object** button serves to deselect objects that are within the marquee frame. To exclude an object, select it with the cursor. The next object selection puts it back in the selected group.



You cannot add an object that is outside of the marquee frame to the selected one.

While the **Exclude/Add Object** button is pressed, the fields of the Property bar are inaccessible. When done with excluding or hiding objects, release the button in order to proceed with the deformation.

3.4.5.2. Deformation by Move



To perform deformation of objects by move, call the **Deformation by Move** command.

Specify the objects to be deformed.

- ▼ If the post-move location of any point of the image is known, set it up as a base point (**p1**). Then set a new location for this point— **p2**.
- ▼ If the moves of the objects along the axes of the current coordinate system are known, enter them in the corresponding fields on the Property bar.
The deformation of objects by move is performed according to the following rule.
- ▼ The elements, which are *entirely inside* the marquee frame, will be moved at a preset distance.
- ▼ The elements that are *partially inside* the marquee frame are edited in such a way that their defining points, which are inside the marquee frame, move at the specified distance, while the defining points that are outside the marquee frame remain in their position.
- ▼ The elements that *are not within* the marquee frame are not edited.

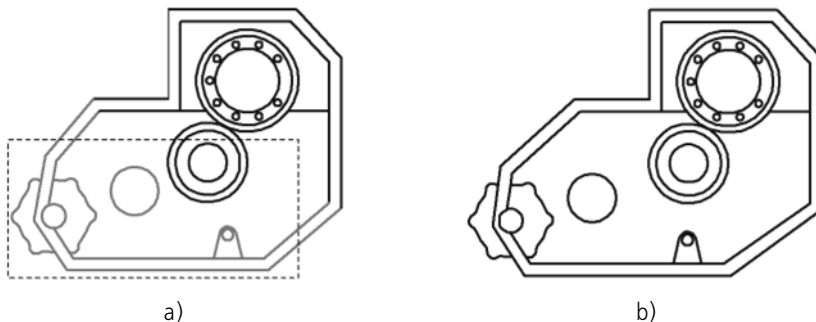


Fig. 3.4.25. Performing a deformation by move:
a) selection of objects, b) result of operation

3.4.5.3. Deformation by Rotation



In order to perform deformation of objects by rotation, call the **Deformation by Rotation** command.

Specify the objects to be deformed.

Select the center point of rotation.

- ▼ If the post-move location of any point of the image is known, set it up as a base point (**p1**). Then set a new location for this point— **p2**.

- ▼ If the rotation angle of the objects is known, enter it in the corresponding field of the Property bar.
The deformation of objects by rotation is performed according to the following rule.
- ▼ The elements that are *entirely inside* the marquee frame will be simply rotated by the specified angle relative to the rotation center.
- ▼ The elements that are *partially inside* the marquee frame will be edited so that their defining points, which are inside the marquee frame, turn by the specified angle relative to the rotation center, while the defining points that are outside the marquee frame remain in the same position.
- ▼ The elements that *are not within* the marquee frame are not edited.

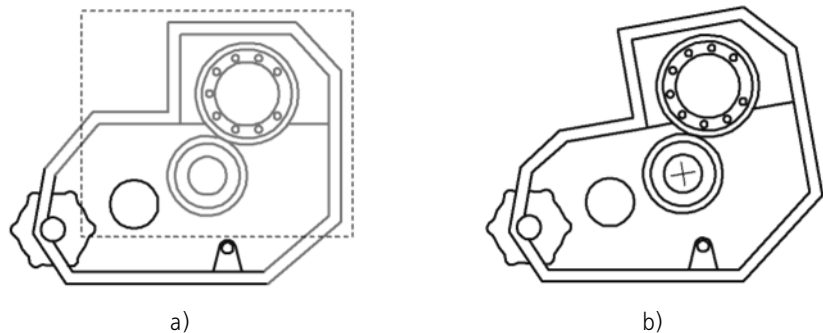


Fig. 3.4.26. Performing deformation by rotation:
a) selection of objects, b) result of operation

3.4.5.4. Deformation by Scaling



In order to perform deformation of objects by scaling, call the **Deformation by Scaling** command.

Specify the objects to be deformed.

Enter in the corresponding fields of the panel the values of the scaling coefficients in the direction of the coordinate axes (you can enter different values of the scaling coefficients along the axes).



You cannot enter the Y-axis scale if there are circles or arcs among the selected objects. In this case, the operation is executed with the scale value along the Y-axis equal to the scale along the X-axis.

Select the center point of the scaling.

After that, the objects will be rebuilt.

The deformation of objects by scaling is performed according to the following rule.

- ▼ The elements that are *entirely inside* the marquee frame will be simply scaled with the specified factor relative to the scaling center.

- ▼ The elements, which are *partially within* the marquee frame, will be edited so that to have the coordinates of their defining points, which are within the marquee frame, modified relative to the scaling center according to the defined factors, while having the defining points, which were left outside the marquee frame, stay in the original position.
- ▼ The elements that *are not within* the marquee frame are not edited.

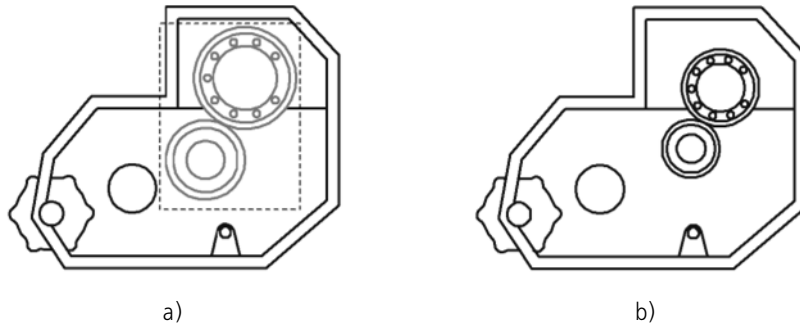


Fig. 3.4.27. Performing deformation by scaling:
a) selection of objects, b) result of operation

3.4.6. Breaking objects into parts

The Break commands can be used for any curves except for equidistant lines and construction direct lines.



If the points specified in the execution of the commands do not belong to the selected curves, the location of the breaking points will be determined by projections of the specified points to the curve. Use snaps for precise positioning of the cursor (see Section 3.1.1 on p. 865).

3.4.6.1. Break a Curve into Two Parts



To break an object into two parts at a certain point, call the **Break Curve** command.

Select a line.

If a curve is not closed, you must specify one point **p1** to break it into two parts.

If a curve is closed, you must specify points **p1** and **p2** to break it.

3.4.6.2. Break a Curve into Several Equal Parts



In order to break an object into several equal parts, call the **Break Curve into N Parts** command.

In the respective field on the Property bar enter the required number of parts, into which the curve must be divided. Then, with the cursor select the line to be divided.

If the curve is closed, then you need to specify the starting point of dividing.

3.4.7. Deleting parts of objects. Extending Objects

Sometimes while editing a drawing you need to delete not an entire element, but only some part of it or, on the contrary, to build up a missing part of such an element. In these cases it is convenient to use special object trimming commands and commands for deleting an area, a chamfer/fillet, commands for aligning objects along border and commands for extension of objects.

3.4.7.1. Curve Trimming



To delete an object part limited by points of intersection with other objects (trim object), call the **Trim Curve** command.

You can trim any geometrical objects, except for equidistants and construction lines.



By default, the piece of the curve that was selected with the cursor is deleted. In this case, the **Delete Selected Portion** switch is active in the **Mode** group on the Property bar. If, on the other hand, you need to delete the outer curve portions relative to the selected one, activate the switch **Retain the Selected Portion**. So, Fig.3.4.28, b shows that to trim both lines, their respective sections, which were located inside a circle, were selected. But the upper line was selected in the mode of saving the specified section, and the lower line— in the deleting mode.



After setting the required mode, specify the required curve section.

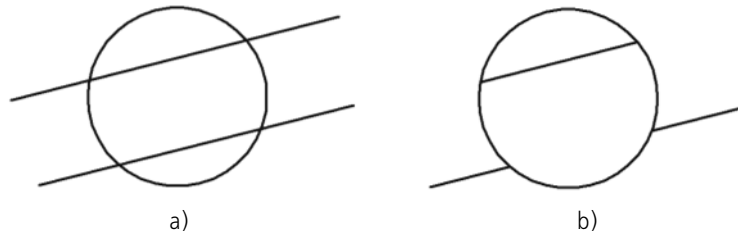


Fig. 3.4.28. Trimming line segments: a) original image, b) result of the command

You can specify a group of objects to be trimmed using a cutting segment.

Specifying a group of objects with a cutting segment

Do the following to specify a group of objects with a cutting segment.

1. Place the cursor at the first point of a cutting segment.
2. Click the left mouse button and move the cursor to the second segment point while holding the button. A cutting segment is forming as the cursor is moving. It is displayed as a dotted line. The objects the segment intersects are highlighted.
3. Release the button after the segment intersects all the required objects.

Figure 3.4.29 illustrates an example how to specify a group of objects to align along the specified border.

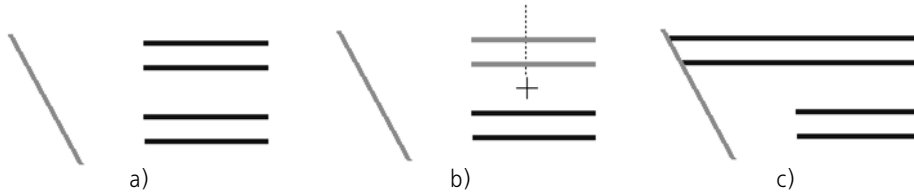


Fig. 3.4.29. Specifying a group of objects with a cutting segment to align along the specified border: a) original image, b) specifying objects with a cutting segment, c) alignment result

3.4.7.2. Trimming Curves at Specified Points



In order to trim an object part limited by two arbitrarily selected points, call the **Trim Curve by Two Points** command.

You can trim by points any geometrical objects, except for equidistants and construction lines.



By default, the curve portion lying between the specified points (if the object is opened) or the portion selected with the cursor (if the object is closed) is deleted. In this case, in the group **Mode** on the Property bar, the switch **Cut Selected Portion** is active. If, on the other hand, you need to delete the outer curve segments with respect to the selected one, activate the **Retain the Selected Portion** switch.



After setting the desired mode, select with the cursor the geometrical object to trim.

Then, specify two points (**p1** and **p2**) that bound the curve segment to delete.



If the specified point does not belong to the selected curve, then the position of the region boundaries will be determined by the mapping of the specified points onto the curve. Use snaps for precise positioning of the cursor (see Section 3.1.1 on p. 865).

If the object is closed, you need to specify a point inside the piece being deleted.

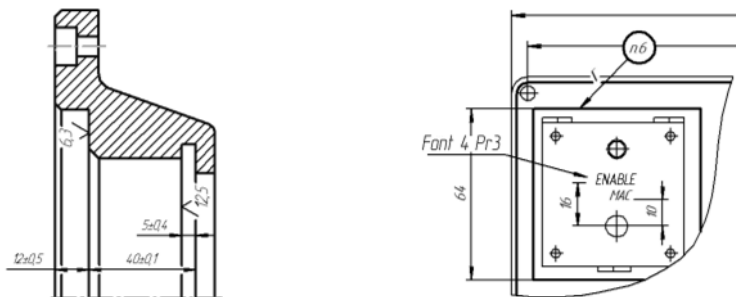


Fig. 3.4.30. Trimming objects by two points

3.4.7.3. Alignment to Border

Alignment of objects along a border is extending objects up to an aligning border or trimming objects along this border.

Such alignment may be required when creating images of bodies of rotation, for example, as on the Fig.3.4.31, and in many other cases.

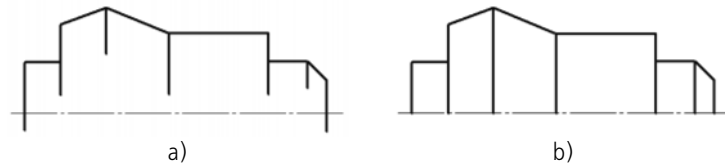


Fig. 3.4.31. Execution of the alignment: a) source image, b) result of the operation

Any geometric objects may be used as a border, and any geometric objects, except for construction lines, may be aligned along the border.



Bezier curves and NURBS can only be trimmed along a border, their extension by using the alignment command is impossible.



To align objects, apply the **Align to Border** command.

Specify the aligning border.

Then select the objects that need to be aligned.

You can specify the objects to be aligned one by one or in a group. Use a cutting segment to specify a group of objects (see Section *Specifying a group of objects with a cutting segment* on p. 1137).

If an object intersects the alignment boundary several times, then all intersections are counted (for example, as in Fig.3.4.32).



Fig. 3.4.32. Execution of the alignment: a) source image, b) result of the operation



To proceed to alignment along another border, press the **Select Other** button and select a new border.

3.4.7.4. Extending to next object



To extend an object up to the nearest point of its intersection (or tangency) with another object, call the **Extend to Next Object** command.

Select the object to be extended, i.e. a line, a circular arc or an elliptical arc.

The extension of an object is performed from its end point, which was the nearest to a cursor when an object was selected, to the nearest point of intersection with another object (including if they are integral parts of macro elements and insertions).

When extending objects, take into account their intersections with geometric objects, axes and lines of break. If none of these objects crosses the extension of the object selected for extension, the extension does not take place.



Fig. 3.4.33. Initial image and image after line segments extension

You can specify the objects to be extended one by one or by a group. Use a cutting segment to specify a group of objects (see Section *Specifying a group of objects with a cutting segment* on p. 1137).

Applying the command once, you can extend several objects or extend one object several times.



If there are several objects, some of which should be extended up to intersection with the object, and some of which should be trimmed in intersection points, use the **Align along the Boundary** command.

Please note that object extension using this command is possible up to visible points of intersection with the boundary and up to points of intersection with its extension while the **Extend up to the Nearest Object** command extends the object only up to the visible points of intersection with other objects.

3.4.7.5. Deleting Chamfers and Fillets



In order to delete a line segment or a curve connecting the ends of two other objects and to extend these objects until they intersect, call the **Delete Chamfer/Fillet** command.

Select with the cursor the chamfer or fillet to be deleted. If the objects connected by the specified line segment or arc can be altered by extending them to the intersection point, then the chamfer or fillet will be deleted.

3.4.7.6. Clearing an Area

When developing drawing documentation (in particular, assembly drawings), the part images may often overlap each other. Naturally, the invisible lines of the part outlines must not be displayed in a document. But deleting them one by one is a time-consuming and tiresome work, which you can automate and considerably speed up using the **Clear Area** command.



So, to delete all the objects located inside or outside a border, call the **Clear Area** command.

The system waits for selecting the borders of areas to be cleared.

If a document contains closed geometric objects (circles, polygons, outlines, etc.) and the image inside them must be deleted, select these geometric objects (Fig.3.4.34).

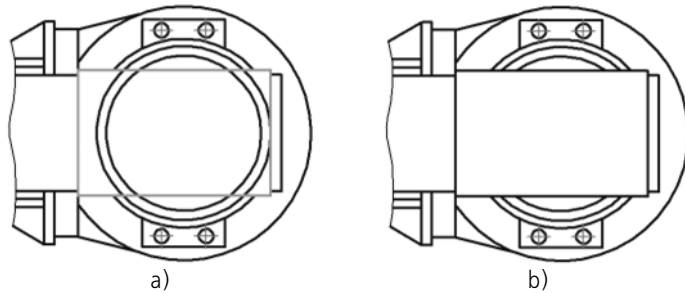


Fig. 3.4.34. Using a closed geometric object as a border for clearing an area: a) selection of an object (highlighted by color), b) result of operation



If the border of the area to be cleared consists of sections of several crossing curves (Fig.3.4.35), select it by following the arrow direction. To do this, press the **Border along Direction Arrow** button. The further sequence of operations is described in Section 3.2.13.1.2 on p. 957.

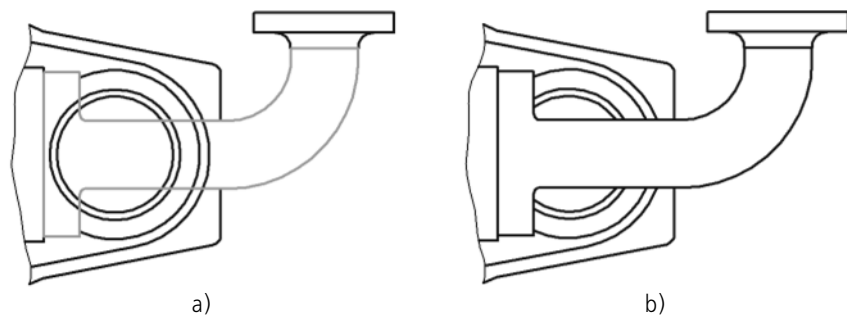


Fig. 3.4.35. Border for area clearing consisting of portions of several objects: a) selection of objects (highlighted by color), b) result of operation



If a border of the area to be cleared is a polyline, which does not coincide with any of the available outlines (Fig.3.4.36), create it manually. To do it, click the **Manual Drawing of Borders** button. The further sequence of operations is described in Section 3.2.13.1.1 on p. 956.

Several regions with boundaries that were defined by various means, located in different positions, can be simultaneously cleaned.



By default, the geometric objects located inside the specified border are deleted. In this case, in the **Mode** group on the Property bar the **Delete Objects inside Boundaries** switch is active. If you need to keep these objects unchanged and delete all other objects, activate the switch **Delete Objects outside Boundaries**.



After specifying all area borders, press the **Create Object** button to clear these areas.



Applying the command for clearing an area, note the following particularities.

- ▼ When the command is performed, the objects, which are located in the current view on the current and the active layers, are deleted (trimmed). For views and layers see Section 3.5.4–3.5.6.

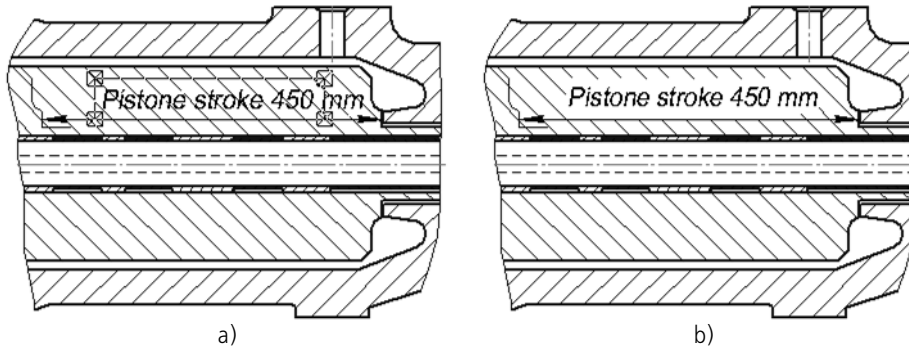


Fig. 3.4.36. Use of a temporary polyline as a border of the area to be cleared: a) creating a polyline, b) result of the operation

- ▼ If among the objects that have partially entered the region to be cleared there is an equidistant, then it will be deleted as a whole.
- ▼ If among the objects that have partially entered the region to be cleared there are layout objects (leaders, datum symbols, geometrical tolerances, etc.), then in most cases those are deleted entirely. The dimensions are special: they are not deleted if there is a dimension text in the region to be cleared. Due to this, you can get the images like those shown on the Fig.3.4.36.
- ▼ The auxiliary straight lines that enter a region to be cleared are left unchanged.

3.4.7.7. Deleting Parts of Layout Objects and Library Macro Elements

The commands for trimming curves, aligning to border and clearing an area may be applied to dimensions and designations and to the images inserted from libraries. In Fig. 3.4.37 the examples of editing some of the specified objects are presented.

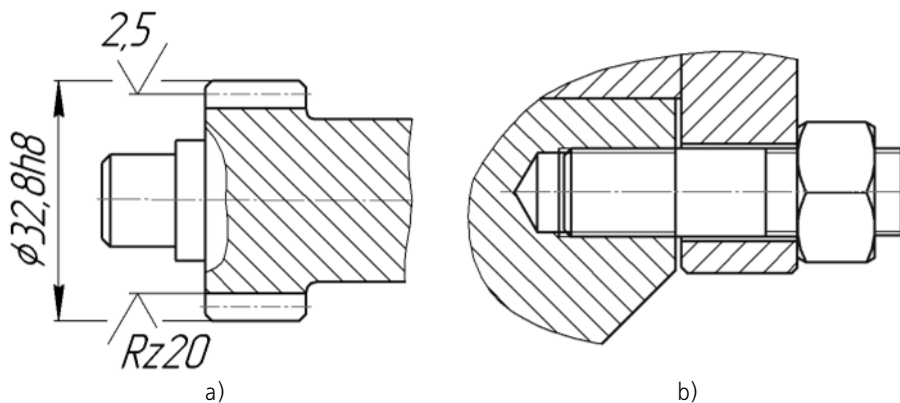


Fig. 3.4.37. Deleting a part of an image:
a) of a dimension extension line, b) of a fastening element (a pin)

But you must take into account that the geometric objects (or parts thereof) deleted in such a manner become invisible only for a while. After the first modification, a layout object or a library macro element will be entirely displayed again.

3.4.7.8. Clearing Background

According to the standard, in case of insufficient space for arrows and texts, you can interrupt outline lines, extension lines, center lines, axial lines and hatches.

As it was described above, you can do it using the commands **Trim by Two Points** (Fig.3.4.30, 3.4.37,a) and **Clear Area** (Fig.3.4.35).

But it is much more convenient to use the background clearing option, i.e. auto interrupting of hatches and lines when they cross the following objects:

- ▼ text on drawing,
- ▼ Arrows, texts and symbols being parts of layout objects (dimensions, leaders, geometric tolerances, etc.).

Enabling and setup of background cleaning in the current document are made in the dialog for setup of overlapping objects' drawing (see Section 9.2.6.17 on p. 1819).

Settings made in this dialog will be applied to all existing and new texts, dimensions and designations created in the current document.

For each of them you can deactivate the background clearing option by calling the **Clear Background** command from the context menu (Fig.3.4.38) or from the **Service** menu.

If the clearing option is deactivated in a current document, this command enables the clearing of the background around the required objects.

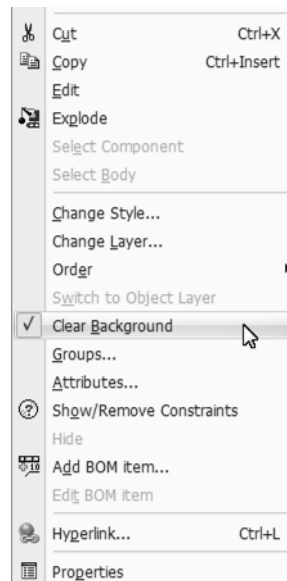


Fig. 3.4.38. **Clear Background** Command



If you individually applied the **Clear Background** command to an object, you can no longer apply the same command settings to this object in the Overlapping Objects Display Customizing dialog.

3.4.8. Deleting Objects

Apart from the commands for entering and editing objects, KOMPAS-3D has a wide range of deleting tools.

The simplest method to delete an object is to select it and press the *<Delete>* button. Selection of geometric objects is described in Section 3.1.3 on p. 876, selection of views is described in Section 3.5.5.4 on p. 1173.



You can delete an object that has just been created by calling the **Undo** command upon its creation.



Be careful when selecting and then deleting macro elements, groups, inserted fragments and other complex objects. Some of objects included in them (for example, those in hidden layers — see Section 3.5.6.1 on p. 1181) may not be displayed, therefore, you can accidentally delete essential elements.

In case you made an error when deleting objects, call the undo command.

3.4.8.1. Deleting the Auxiliary Objects

While working with a drawing, the designer often uses various auxiliary constructions (just like thin-line construction on a drawing board).

KOMPAS-3D offers various options for creating infinite straight line. The line style used for them — *Auxiliary* — cannot be changed. You can set this style for any geometric object during its creation or editing. In addition, the style *Construction* can have points (this very style is used by default for points).

When editing a fragment, you can delete its auxiliary constructions elements, which became unnecessary, by calling the **Edit— Delete— Construction Lines and Points** command. All curves and points having a *Construction* style will be deleted from the fragment.

After the command **Edit— Delete— Construction Lines and Points** is applied during editing a drawing, a submenu, which includes the commands **In Current View** and **In All Views**, appears on the screen. In such a way you can delete the objects of a construction style only from the current view or from all views of a drawing simultaneously.

3.4.8.2. Deleting All Document Objects

To delete all contents of a document at once, call the command **Edit— Delete— All** command. After that, a screen will display a warning that the cancellation of an operation is impossible. To confirm the deletion, press the **Yes** button.

If the deletion of all objects of a document was in error, the only way to restore the contents contained in a document, following the previous saving on a disk, is to close this document without saving it on a disk and open it again.

3.4.8.3. Deleting Layout Objects

To delete such drawing layout objects as a title block, BOMs and an unspecified surface finish symbol, apply the following commands from the **Edit—Delete** menu: **Title Block Content**, **Technical specifications**, **Unspecified Surface Finish**.

It is impossible to restore the contents of a title block (data entered manually into the title block cells). Therefore, after you call the respective command, a warning dialog will appear on a screen, where you can confirm or cancel the deletion.

3.5. Creation of drawings

3.5.1. The Structure of a Drawing

Besides the graphical image, a drawing contains the a border, the a title block, the a symbol of unspecified surface finish and the specifications. Geometrically, a sheet is characterized by its format. It includes the format itself (A1, A2, etc) as well as multiplicity and orientation.

If a drawing contains several sheets, each of them can may be assigned a separate format and title block type.

3.5.1.1. Pages

Each sheet is displayed in the drawing with an outer and inner format frames and a title block. All sheets of one drawing are displayed on the screen simultaneously. Those are positioned tight next to each other from left to right in the order they were created.

The sheets are in no way related to the image stored in the drawing. Simplifying, those may be assumed as lying on a special layer located above the rest of graphical objects. Therefore, when deleting a sheet, the image that was “underneath it” stays in its original position while the frame around it and the respective title block disappear.

Upon creation of a new drawing, its first sheet is automatically created. More sheets can be added, if necessary. This can be done at any point while working with on a drawing. The parameters of any drawing may be modified at any moment, time as well. Sheets management is described in more detail in Section 3.5.2 on p. 1150.

This serves to enable the zoning of a drawing and to set zone dimensions and their designations. Enabling and setup of a drawing's zoning is described in Section 9.2.6.18.4 on p. 1824.

3.5.1.2. Views

A view is a part of a drawing, containing the image. Graphic objects can be placed on one or more layers in a view (see more about layers in Section 3.5.6 on p. 1181). An image cannot exist without a layer and view assigned.



The above-mentioned does not apply to technical specifications and unspecified surface finish symbol — they do not belong to any view or layer.

The main properties of a view are its **scale** and **position**. Changing the scale or location in a view leads to scaling and moving all objects in this view.

Drawings created using the KOMPAS-3D system can contain up to 2,147,483,647 views.

A drawing view does not have to contain some geometrical projection of a part in strict geometric sense. It may be any isolated image.

A drawing doesn't not have to be divided into views. All graphical information of a drawing can be included in one view, if necessary.

Upon creating a new drawing, a special system view with the zero ID is automatically created in it, and in a view— a system layer with the zero ID is added to the view.

If the user did not create any other views and/or layers, then all objects being entered in a drawing will be placed into the system view on the system layer, and in a fragment — on the system layer.

Thus, you can start making an image right after a drawing is created, not bothering about view creation.



In this case, the drawing will be done in “actual size”. Drawing in scale is described in Section 3.5.4.1 on p. 1161.

Inside a fragment, breaking into views is impossible, because the fragment itself is similar to a system drawing view.

Working with views is described in Section 3.5.4 on p. 1160.

3.5.1.3. Symbol of Unspecified Surface Finish

Symbol of unspecified surface finish is almost always present in the engineering part drawings of machine building parts. The KOMPAS-3D system allows a symbol to be created and placed automatically.

Text parameters of the unspecified surface finish symbol cannot be entered during the symbol creation. These parameters are defined by document settings described in Section 9.2.6.18.7 on p. 1827.

3.5.1.3.1. Placing the Symbol

To place an unspecified surface finish symbol on a drawing, select **Insert— Unspecified Surface Finish— Enter...**

A dialog window for adding and editing the symbol will open (Fig. 3.5.1). Its interface elements are described in Table 3.5.1.

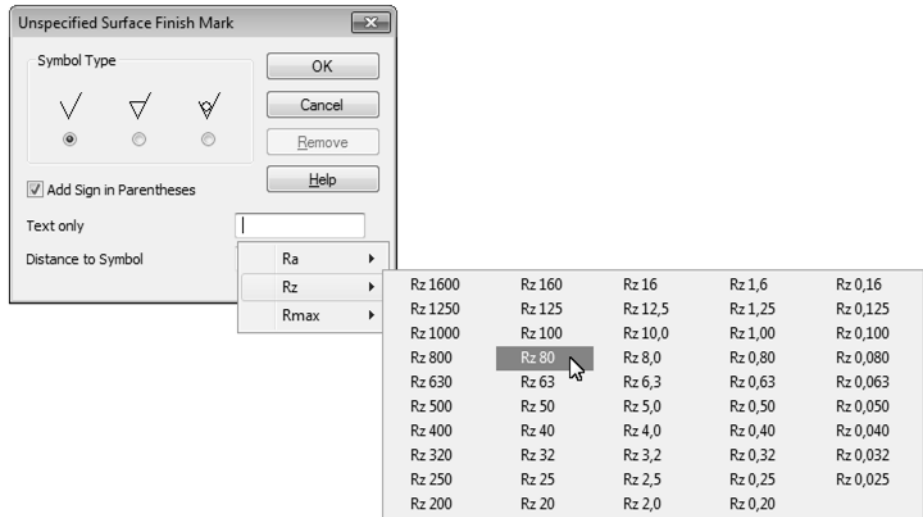


Fig. 3.5.1. Dialog for adding and editing the symbol

Tab. 3.5.1. A dialog for adding and editing the Unspecified Surface Finish symbol

Element	Description
Symbol type	A group of options making it possible to select a line type: Without Machining Type, With Material Removal or Without Material Removal.
Add Symbol In Parentheses	An option for drawing the symbol in parentheses.
Text	Label Text. It can be typed in from the keyboard, or selected from the user menu (Fig. 3.5.1). To open the user menu, double-click the field to be populated (this differs from opening the context menu which is also available for this field).
Distance to the Symbol*	The field for entering the vertical spacing between the title text and the surface finish symbol.
Delete	The button allowing to delete the symbol. The button becomes enabled after a symbol is created.

* This field is in the dialog if, when setting up the surface finish symbol, the correspondence to the previous edition of GOST2.309-73 was selected. This setup is made in the **Surface finish** section in the current document setup dialog (see Section3.3.7.2 on p. 1019).

After the dialog box is closed by clicking OK, the symbol appears in the upper right corner of the drawing sheet.

3.5.1.3.2. Editing and Deleting the Symbol

The unspecified surface finish symbol can be changed using the symbol editing dialog box (Figure 3.5.1). To open the dialog box, do one of the following:

- ▼ double-click the symbol,
- ▼ select **Edit Unspecified. Surface Finish in the context menu**,
- ▼ select **Insert— Unspecified Surface Finish— Enter...** in the main menu.

To change the location of the symbol on the drawing, enter the symbol locating mode. Do one of the following:

- ▼ select **Manual Location** in the symbol context menu
- ▼ select **Insert— Unspecified Surface Finish— Location**.

The symbol boundary will be displayed, and its anchor point highlighted. The symbol can be dragged with the mouse by a defining point to any position on the drawing.

To restore the default location of the symbol¹, select **Auto Locate** in its context menu.

To delete an unspecified surface finish symbol, do one of the following:

- ▼ click the **Delete** button in the symbol editing dialog box (Table 3.5.1 on p. 1149).
- ▼ select **Delete Unspecified. Surface Finish** in its context menu
- ▼ select **Edit — Delete — Unspecified Surface Finish** in the main menu.

3.5.1.4. Technical Specifications

The specifications are an integral part of a drawing. They supplement graphical information contained in a document.

Technical specifications represent text on the drawing's margin that belongs neither to views, nor to layers. They may contain links to text objects of the current drawing and be sources of links.

Manipulation with technical specifications in a drawing – creation, editing, formatting and other operations – is performed in the *mode of specifications handling* (see Section 3.5.7.1.1 on p. 1199).

Locating the specifications on a drawing, breaking it down into pages, editing and deleting of a drawing are described in Section 3.5.7 on p. 1198.

You may set up the document specification parameters that will be used by default. Setup of specifications' parameters is described in Sections 9.2.6.18.5 on p. 1826 and 9.2.6.18.6 on p. 1826.

3.5.2. Sheet

The main instrument intended for sheet management is the **Document manager**.

To open the **Document Manager**, do one of the following:

-
1. By default, the Unspecified Surface Finish symbol is placed in the upper-right corner of the first drawing page at the distance of 7 mm from the top frame boundary and 8 mm from the right frame boundary.

- ▼ select **Service— Document Manager** in the main menu
- ▼ click the **Manage Views** button in the **Current Status** tab
- ▼ click the **Layer Management** button in the **Current Status** tab,
- ▼ select **Insert— Layer....**

A detailed description of the **Document manager** interface and ways to work with it is provided in Section 3.5.6.2 on p. 1182. This section describes ways of working with views using only the **Document manager**.

You can work with additional sheets of BOM in the Document manager in the same way as you work with sheets of a drawing (see Sections 3.5.2.1–3.5.2.3).

3.5.2.1. The Title Block and the Sheet Format

The title block table makes one of the elements of the sheet **layout**. A layout also includes external and internal frames. Layouts supplied with KOMPAS-3D are stored in libraries, i.e. in *.lyt files located in \Sys sub-catalogue of the main system catalogue. The basic library of layouts used when creating documents is *graphic.lyt*. It is also possible to form user designs libraries.

A template assigned to each sheet allows you to refrain from drawing the borders and tables of the title block.

Completing Filling the title block in is described in detail in Section 3.5.3.1 on p. 1155.

By default, the first sheet of a drawing created without a template (see Section 10.2.1.8 on p. 1982 for more information about templates), has the following layout: *Design drawing. First Sheet. GOST 2.104-2006*, while newly added sheets have the following template: *Design Drawing. Other Sheets. GOST 2.104-2006*. Default sheet format — A4.

If a drawing is based on a template, the number of sheets and their layout are determined by the template.

Regardless of the document creation mode, you can change the layout and format of any sheet.

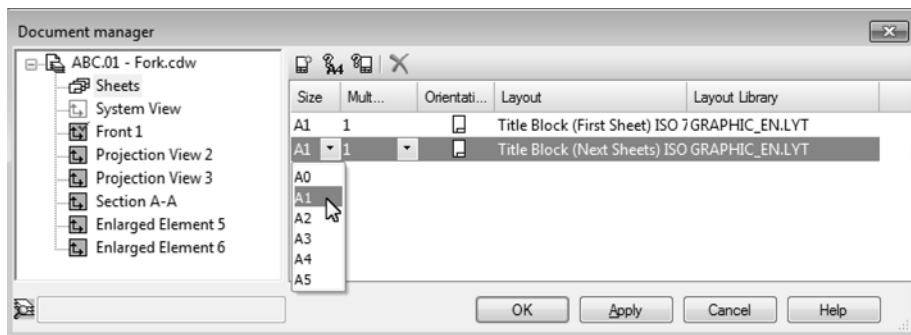


Fig. 3.5.2. Document manager; working with drawing sheets

For this, do the following.

1. Call the **Document manager**.
2. Activate the **Sheets** element in the Tree of Sheets, Views and Layers **of the Document Manager**.

The **Sheets, Views and Layers List** shows all the sheets of a drawing in the order they were created. The properties of the sheets are shown, as well (Figure3.5.2).

3. Click the line representing the sheet in the list to change the parameters need to be changed.
4. An icon in the **Orientation** column shows the current orientation of the sheet. To modify it, click the respective sign with the mouse. The icon will change as well.
5. The **Format** column shows the current format of the sheet. To change the format, open the drop-down list menu in this column and select an appropriate format.



You can set a non-standard sheet size. Select **Format** in this line's context menu, or click the **Format** button in the toolbar of the **Document Manager**. When a dialog box opens, set the **User** option, enter sheet size and close the dialog box. The **Format** column will show the selected sheet format.

6. The **Multiplicity** column will show the current sheet multiplicity. To change the multiplicity, open the drop-down list and select new value.



In case of a non-standard format, the multiplicity cannot be set.

7. The **Template** column shows the name of the layout assigned to the sheet from the current template library. This library filename is shown in the **Template Library**.

7.1. To select another template from current library, click the value in the **Template** column. Specify the required template in the dialog box that opens.

7.2. To select a template from another library, click the value in the **Template Library** column and click the **Template** button in the **Document Manager** toolbar. Specify the required library and template in the dialog box that will open.



8. The changes can be viewed on screen without closing the dialog box. To do so, click **Apply**. To close the **Document Manager**, save changes and continue working with a drawing, click **OK**.



The first drawing sheet properties can be set up by another method. Call the command **Service— Options** or the **Settings of Current Drawing...** command from the document window context menu. Open up the **Current Drawing** tab of the displayed dialog and perform the desired settings using the interface items of the **First Page Parameters** Section.

If the same template and format are used for most of the drawings, there is no sense in setting them up for each document individually. In this case, you can create all the new documents with the required parameters from the beginning.

To do this, call the **Service — Parameters... — New documents — Graphic document — Parameters of the first sheet/Parameters of new sheets**.

To set these sheet parameters for all future drawings, select the **Format** and **Template** in the left side of the dialog box.

For details on drawings' layout, see Section 10.2.1 on p. 1959.

In **Document manager**, you can set the layout and other sheet parameters for BOMs.

To setup additional sheets of a specification, perform steps described in paragraphs 1-8 for each of them. When performing steps from paragraph 2, enable the **Additional sheets in the beginning/end of document** object in the **List of sheets, views, and layers**.

To set up a BOM style, activate the **Document Sheets** object and take the steps described in 7–8.

3.5.2.2. Adding a Sheet

There are two ways to add a new sheet into a drawing.

- ▼ Using the Main Menu. Call the **Insert— Sheet** command. A new sheet will appear on the drawing to the right of the existing sheets. Its layout and format will be determined by the settings made for new drawing sheets. If necessary, those properties can be modified (see Section 3.5.2.1).
- ▼ By using **Document Manager**. For this, do the following.
 1. Call the **Document manager**.
 2. Select the **Sheets** object in the Tree of Sheets, Views and Layers.
 3. Click the **Sheet creation** button on **Document Manager** toolbar. A line corresponding to the created sheet will be added at the end of the List of Sheets, Views and Layers, which corresponds to the created sheet. If necessary, you can change the sheet properties.



If you need a sheet to be added to a certain position, specify the sheet in the List of sheets, views and layers, that should be followed by the inserted sheet, before you click the **Create sheet** button.

4. Click the **OK** button in the dialog **Document Manager**. It will be closed, and the added sheet will appear on the screen at the right of the existing sheets.

In **Document manager**, you can add additional sheets in the beginning and in the end of a BOM.

3.5.2.3. Deleting a Sheet

To delete a sheet from a drawing, do the following:

1. Call the **Document manager**.
2. Select the **Sheets** object in the Tree of Sheets, Views and Layers.
3. In the Sheets, Views and Layers List, select the sheet to be deleted and click the **Delete** button.
4. Click the **OK** button in the dialog **Document Manager**. The dialog box will close, and the deleted list will disappear.



A drawing cannot exist without sheets. Thus, it is impossible to delete the last sheet in a drawing.

In **Document manager**, you can delete additional sheets in the beginning and in the end of a BOM.

3.5.2.4. Switching between Views

All sheets of a multipage drawing are displayed on the screen simultaneously. However, while drawing the objects, the scale is usually set to display only one sheet or part of a sheet. In so doing, all the rest screen's scope.

A sheet that is located in the center of the document window (excluding the Drawing Tree) is considered **current**. Its number is displayed in the list of sheets on the **Pages Management** bar (Figure 3.5.3).

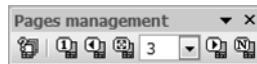








Fig. 3.5.3. **Sheets Management** panel

To move to another sheet, use the scrollbars, the **Move** control, or the **Sheet management** panel. The elements of this panel are described in Table 3.5.2.

Tab. 3.5.2. **Sheets Management** panel

Element	Function
 Library of a document	Open the Document Manager dialog box.
 First Sheet	Serves to make the first document sheet current. Display scale does not change. The button is inaccessible, if the first sheet is current.
 Previous Sheet	Serves to make current the sheet preceding the one that is current at the moment. Display scale does not change. The button is inaccessible, if the first sheet is current.
 Show Sheet	Show the current sheet in full. In this case, the display scale is adjusted to fully fit the current sheet in the window drawing, and the current sheet position is changed to make its center aligned with the drawing window center.
List of Sheets	Select the current sheet. Expand the list and select in it the desired sheet number. The sheet with the selected number will be displayed on the screen without scale change.
 Next Sheet	Serves to make current the sheet following after the one that is current at the moment. Display scale does not change. The button is inaccessible, if the last sheet is current.

Tab. 3.5.2. **Sheets Management** panel

Element	Function
 Last Sheet	Serves to make the last document sheet current. Display scale does not change. The button is inaccessible, if the last sheet is current.

3.5.3. The Title Block of a Drawing

The title block is automatically placed onto the sheets of a drawing, while the user just has to fill in the cell contents. Some cells allow semi-automatic text entry.

Since the title block is a part of a template, the user cannot change its size and structure from inside the document.

To place another title block onto a document sheet, another template containing the required title block should be assigned to this sheet document, containing the required title block.

If a sheet requires a non-standard title block, you have to design this title block beforehand, and include it into a template and then assign this template to the document.

3.5.3.1. Filling title block

Filling in the title block is, in general, similar to entering text into the table's cells.



Standard cells, like *Designed by*, *Checked by*, etc cannot be populated or edited.

There are three ways of entering the mode for filling in the title block:

- ▼ double-click the title block,
- ▼ select **Fill in title block** in its context menu,
- ▼ select **Insert— Title block** in the main menu.

The title block changes its appearance when it is in the fill-in mode— cell boundaries show the text margins.

Enter or edit the text in the title block.

The system allows to fill in the title block semi-automatically. After double-clicking in a title block's cell, the following will appear on the screen: a dialog for designation entry or a menu, from which the required line can be selected, or the calendar, from which you can choose the date, or the text template selection window. Document codes and names can be selected from a special dialog box.

These functions are described below in more details.

The *Scale* cell can be populated by a reference to the scale of one of the existing views of the drawing (see Section 4.1.4.3.1 on p. 1338 about reference creation). When the scale in this view is changed, the value in the *Scale* cell will be updated.



The title block can be set up to contain the link to the first view created by the user. Enable the **Create reference to view scale in title block check box** in the new view settings dialog box (see Section 9.2.6.18.1 on p. 1820).

After all title block cells have been filled in, click the **Create object** button or press **<Ctrl>+<Enter>** keys on the keyboard to save changes and exit the title block edit mode.

The data entered into cells of the **Common for document** type is automatically transferred to other cells of the same type (this includes other sheets, as well). Cell types are preset during setup of title blocks included into page designs. For example, in the templates included in the installation package, the product designation entered in the title block on any sheet is copied to the *Cells 26* and *Designation cells* in all other sheets.

Individual information is to be entered at each page.

Number of pages and *Page number* columns are filled in automatically and are not available for editing. If necessary, you can set a predefined number of sheets and disable auto-detection of the sheet number (see Section 9.2.6.18.3 on p. 1824).

3.5.3.1.1. Document designation

The document designation can be **simple** or **complex**. Designation is considered simple if it includes only basic designation. A complex designation includes, apart from basic one, version number, and/or additional version number, and/or document code.

You can input or edit simple designation directly in the cell of the title block, while working with complex designation requires a special dialog (Fig. 3.5.4), with a separate field for each part of the designation.

Base designation	-	Ver. number	-	Add. number	-	Code
AKPM.01001	-		.			BV

OK Cancel Help

Fig. 3.5.4. Designation entry dialog

In order to call the dialog for working with the complex designation, use the **Edit designation...** command in the context menu.

Enter or edit the required designation elements in this dialog. You can only enter one symbol in the separator field. One line can be entered in any other field.

The dialog's context menu contains the main commands for working with text. The same or other commands are available in the main menu and in the Property bar tabs.

You can specify the document's code by direct entry or by selecting it from the **Codes and names** dialog (see Section 3.5.3.1.5 on p. 1158).

Having finished designation entry or editing, close the dialog by clicking **OK**. The designation composed of fragments that are specified in the designation entry dialog fields will appear in

the title block. If the document's code was specified in the dialog, the respective code's name will be added to the *Name* field.



System separators (dash – the first one, point – the second one) are not included in the ready designation, if the respective part of the designation (the variant's number or additional number) is not specified.

If other symbols were entered instead of system separators, they are included in the designation without any prerequisites.

If the designation is a complex one, the **Designation** dialog will appear automatically when you try to edit it. Besides that, you can also double-click the designation in order to call the dialog.



Parts of a composite designation should be presented in different files and, respectively, the **Designation** dialog should be used if you plan to work with BOM and reports because such presentation provides adequate sorting. Otherwise, all parts of a composite designation can be entered as one line (which will be treated as a basic designation).

A composite designation “turns” into a simple one if fields with the version number, the additional number and the code (delimiters should be of system type or should not be present) are cleared in the dialog.

3.5.3.1.2. User Menu

A user menu opens upon double-clicking any cell requiring a surname to be entered. It includes the **Developers**, **Production managers** and **Standards Quality control** options, each of which shows a the list of surnames (Figure 3.5.5). The surname selected in one of these lists, is automatically entered into the corresponding cell of the title block. If the surname is too long, the character width will be adjusted automatically to fit into the cell margins.

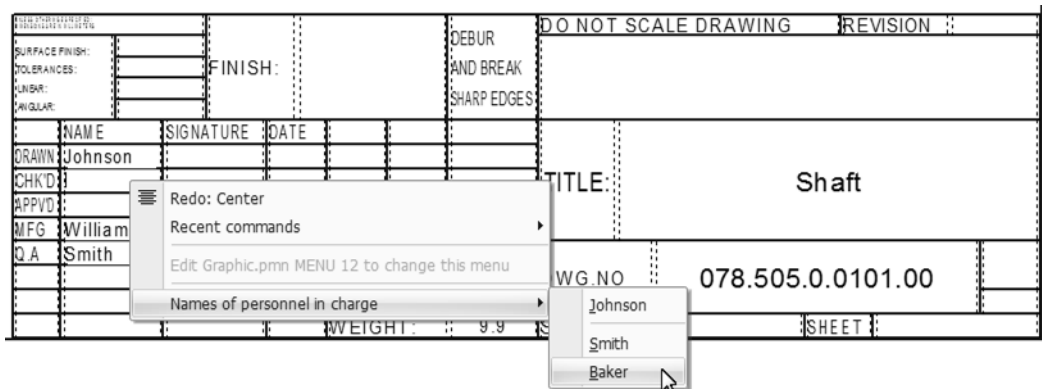


Fig. 3.5.5. An example of selecting the surname of a standards quality control staff member for entering into the title block

The menu structure and contents shown in Figure 3.5.5 are stored in the user menu file called *Graphic.pmn*. This file is included in the installation kit and is located in the \Sys subfolder of the main KOMPAS-3D program folder. You can modify this file by adding more menu items or

sections, or by adding surnames to existing sections. The changes will be displayed when the menu is opened the next time.

Double-clicking the *Scale* and *Codes* cells opens context menus as well. Content of these menus is also stored in the *Graphic.pmn* file, and allows user updates.

3.5.3.1.3. Date

Double-clicking any date cell opens the **Enter date** dialog box (Figure 3.5.6). By default, the current date is active. You can select another date by using the list of years, months and days. After pressing the <Enter> key, the specified date will be automatically placed into the appropriate entry in the stamp. The date is entered in the Windows short date format.



Fig. 3.5.6. Selecting a date for the title block

3.5.3.1.4. Text Templates



Double-clicking other cells of the title block opens the Text Templates Librarian¹. Select a required template and click **Insert to Document**. The text will be automatically entered into the cell. For details on text templates, see Section 4.1.2.11.6 on p. 1312.

If the text to be entered in the cell is not included in the text template list, enter it manually. If you foresee a need to enter the same text several times repeatedly, enable automatic saving of the entered text into the text templates file. For this, do the following.

1. Call command **Service — Parameters.... — System — Text Editor — Text Templates** command.
2. On the right side of the dialog box that opens, check the **Save entered text to file** check box. If necessary, specify the name of the file to save the texts to (by default, the *Graphic.tdp* file is used).
3. Click the **OK** button to apply the settings.

3.5.3.1.5. Codes and Names

The nomenclature of engineering documents is established by GOST 2.102–68. According to the standard, each document type has its certain code and naming. While filling in Title Blocks of the KOMPAS-documents, these codes and names can be entered automatically.

1. If the Materials and Assortment Handbook, or the Materials and Assortment Library are installed on your workstation, then double-clicking a cell containing material data will open the Handbook or the Library, respectively.

To enter code and naming, use the command **Number and name...** It is available in the document's title block entry and when working with the designation entry dialog (see Section 3.5.3.1.1 on p. 1156). The command is available in the **Insert** menu and in the context menu. After calling the **Number and name...** command the **Codes and names** dialog appears on the screen (Fig.3.5.7).

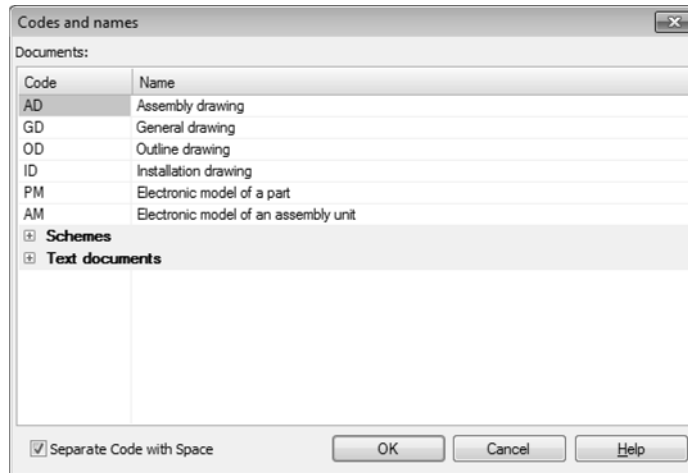


Fig. 3.5.7. **Codes and names** dialog

Document codes and names are listed in this dialog.

A “+” sign is located onto the left of a section name. To expand a section, click the respective sign with the mouse. It will change to “-”. The section documents will appear in the list.

Select the desired code and click the **OK** button. You can also double-click on the «**Code**» entry. The dialog will close. Selected code will be inserted in *Designation* cell, and the name corresponding to it — to *Name* cell. The insertion method depends on the contents of the cells and the following rules.

- ▼ If the entries are empty, then the document code and name are inserted there.
- ▼ If the entries contain an arbitrary text, then the document code and name are inserted after that text.
- ▼ If the last signs in *Designation* cell represent one of standard codes, these signs are deleted. The selected code is inserted instead of them.
- ▼ The last signs in *Name* cell will be deleted if they are one of the standard document names. The selected document name will be inserted instead of them.
- ▼ Any number of dots or space characters can exist after a text string made of standard code or standard name. When the selected code and name are inserted, these characters are deleted.



Codes and names are entered automatically into the cells of the following types: *Product name* and *Document designation*.



Contents of the **Codes and names** dialog is defined by the file of codes and names — *graphic.kds*. By default it is stored in \Sys subfolder of the main system folder. File format is a text format which is why the *graphic.kds* file may be opened and edited in any text editor if necessary (see Section 9.5.5.3 on p. 1899).



Inserting a code from the dialog transforms simple designation into a complex one (see Section 3.5.3.1.1 on p. 1156).

3.5.3.2. Deleting the Contents of the Title Block

This can become necessary, if a new drawing is created from an existing drawing.

There are two ways of deleting the title block contents of the title block:

- ▼ select **Edit – Delete– Content of title block in the main menu**,
- ▼ select **Delete contents** in the title block context menu.



Be very careful while deleting the title block contents, since this operation cannot be canceled.

3.5.4. General information about views

From designer’s standpoint, the view is an image of a visible part of object surface.

A view, as part of a KOMPAS drawing, is a “container” for objects, and includes as well as the objects themselves located in such “container”.

Objects contained in one KOMPAS-3D view may form either a single image (view, cut, section or enlarged detail), or multiple ones. Theoretically, a drawing may consist of a single view which contains all necessary images.

However, we strongly recommend breaking down all graphical information of a KOMPAS-3D drawing into views and place each image into a separate view. This approach provides the following advantages.

- ▼ The image can be displayed in different scales with no need to manually recalculate the dimensions, since they are recalculated automatically (see Section 3.5.4.1).
- ▼ The images can be conveniently positioned on the sheet: The images can be conveniently positioned on the sheet: each view can be scaled, moved and rotated as one object. (see Section 3.5.5.7 on p. 1174).
- ▼ It becomes possible to make an associative relation between the direction arrow designation symbols, cut/section lines, enlarged details and the designation symbols of the respective images. Due to this relation, such data like a letter, page number, etc. are automatically exchanged between the designation symbols.

Such capabilities noticeably speed up the creation of assembly drawings, drawings of large objects and densely populated drawings.

Additionally, you can allow/prohibit editing any views at any moment of working with a drawing (by making them active or background), and also enable/disable views display (by making

them shown or hidden). For details on view states, see Section 3.5.4.2 on p. 1162. You select a desired property for a particular view at the time of setting up a view status (see Section 3.5.5.2 on p. 1171)

You can make one or several breaks in images on any view except for the system ones (see Section 3.5.5.8 on p. 1177).



Associative views represent a special type of views. They contain automatically formed projections of 3D models. Associative views are detailed in Part 3.6.

3.5.4.1. Drawing the Images in Different Scales

When creating KOMPAS-3D drawings, the user may work using the scale 1:1, i.e., when creating geometrical objects (line segments, arcs, etc.) that make up the product's contour, natural dimensions are used, and views can be used to scale the image.

For example, to show a construction 1500 mm long on an A1 page, it should be drawn in scale 1:2.5. In traditional paper drawing each dimension should have been divided by 2.5 to get such a reduced image, moreover, the new values should have been manually added into the dimension texts.

KOMPAS-3D allows creating a view in scaled 1:2.5 immediately (i.e., prior to creating an image) beforehand, and then draw an image in this scale using actual size values. Image scaling (reducing it 2.5 times) will be automatically performed by the system. Actual values of dimensions will be automatically calculated by the system when dimensioning.

If you find in the future that the scale needs to be modified, the image does not need to be redrawn later, recalculating the dimensions. You will just need to change the scale of the view in which this image is located.

When creating a new drawing, a system view with the 1:1 scale is automatically generated and it.

Settings of the system view cannot be changed. Therefore, if you need to create an image in a drawing to a scale other than 1, you first need to create a new view with the desired scale.

If you use the same scale for most of the drawings, then creating such a view in each drawing is inefficient. In this case, new drawings can be set up to already contain a view with the required scale factor. To do this, call the **Service — Parameters... — New Documents — Graphic Document — Document Parameters — View**,

On the right side of the dialog box that opens, check the **Create new view** check box and set the scale of the view. The values can be either entered manually as arbitrary numbers, or selected from a list of standard scale factors that opens on clicking the button between the scale entry fields.

The scale defined by this setting will be automatically propagated to the respective entry of the title block in newly created drawings.



Regardless of the **Create New View** option state, when creating a new drawing, a view automatically generates its new system view in it, called Hand over.

3.5.4.2. View States

The state of a view is defined by the values of the following parameters:

- ▼ **Active,**
- ▼ **Visibility.**

The **Activity** property manages the availability of view objects for editing and can have two states: **active** and **background**.

Objects of the active view are accessible to the for editing and deletion operations. The contents of an active view are displayed in one color selected for this view upon setting it up.

A background view is accessible only for snapping operations (see Section 3.1.1 on p. 865) to points or objects. Such a view cannot be moved, and its objects are not accessible for editing. The contents of all elements of background views are displayed on the screen in the same style that which can be customized (see Section 9.1.7.8 on p. 1710).

The **Visibility** property manages the display of a view and also has two states: **visible** and **hidden**.

If the view is visible, then it is displayed on the screen. In this case, the active views are shown in the colors selected for them, while the background ones are shown using the specified style.

If a view is hidden, then it is not displayed on the screen regardless of whether it is active or background. Therefore, a hidden view is completely inaccessible for any operations.

Among all drawing views, there is one and only one that has the **current** status.

It is the current view (its current layer) that the newly created objects are placed into.

Any view can be made current. In this case it automatically becomes visible and active. These states cannot be changed while the view is still current (i.e., current view cannot be hidden or put into background). Objects of the current layer are displayed in the real styles of lines, points and hatches, which are set in the system settings dialogs.

Once the **current** status is assigned to another view, the state of the view that used to be the current before is restored. For example, at some moment of working with the drawing, View 1 was current. View 2 was both background and visible at the time. Then, the View 2 was made current, so that it became active. After finishing editing View2, View1 becomes current again. View2 became background again at the same time.



Depending on the system settings (see Section 9.1.5.2 on p. 1701), the hidden views can or cannot be printed.

Controlling the view states is described in Section 3.5.5.2 on p. 1171.

3.5.4.3. Creating a Simple View



In order to create a simple (user) view in a drawing, call the **Insert — View** command or click the **Create new view** button in the **Views** bar.



The cursor shape will change to the coordinate system icon.

Define parameters of the new view (see Section 3.5.4.4) and set its anchor point.

3.5.4.4. View Parameters Setup

To set up the view parameters of a view, use the **Settings** tab of the Properties bar. These elements are presented in Table 3.5.3.

Tab. 3.5.3. Interface elements for managing the settings of drawing views

Element	Description
Number*	The field containing the identification number of the view. By default, a new layer is assigned the first unused number. Any number can be entered here, different from the numbers of existing views may be entered here.
Name*	The field containing the name of the view displayed in the Drawing tree (see Section 3.6.1.1 on p. 1207) and in the Current view field in the Current status bar. Any sequence of characters can be entered here.
Color	The color of the view in active state (for details on view states see Section 3.5.4.2 on p. 1162).
Scale	The scale of the view, select from drop-down list. Scale factors can also be entered from the keyboard.
Extension Lines	The group of switches allowing to indicate whether it is necessary to scale the dimensions of the extension lines and leaders (if there are any) in a view when its scale is changed. See Section 3.4.4.2.1 on p. 1131 for more information about the dimensions' extension lines scaling.
View Point	Switches allowing to select the base point for the view.
Snap Point	The coordinates of the view anchor point (that should match the base point of the view) in absolute coordinate system numbers (the point of origin of the drawing absolute coordinate system is in the bottom left corner of its text boundaries).
Angle	Rotation angle of the view around its anchor point. To rotate the view around another point, select the Rotation command (see Section 3.4.4.1 on p. 1129).

* The field is not displayed in the Properties bar when standard associative views are being created (see Section 3.6.2.1 on p. 1216).

3.5.4.5. Drawing Tree

Drawing Tree is a graphically depicted sequence of views creation in the current drawing (Fig. 3.5.8).

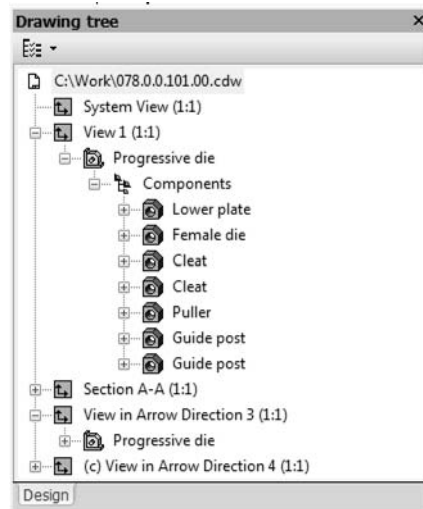


Fig. 3.5.8. Drawing tree

The Drawing Tree window can be shown/hidden by the command **View — Drawing Tree**. When the Tree display is On, a “checkmark” is displayed next to the command name in the menu.

If several windows are open with the same drawing, then the Tree can be shown or hidden in any of them.

Drawing Tree is displayed in a separate window which is always inside the drawing window. You can resize the Tree window by dragging its corners or side borders with the mouse.

Views are displayed in the drawing tree in the order of their creation.

The “+” icon may be displayed to the left of the view name in the Tree. Click this icon to display objects linked to the view — macroelements, view and fragment insertions, and also the model for the associative view the image of which is contained in the view, as well as local cuts (if there are any in the view). More information about associative views in the drawing tree can be found in Section 3.6.1 on p. 1205.

Each view automatically appears in the Tree right after it was created in the drawing. The name is also assigned automatically to the views. It contains the view name and its scale. For example, “System View (1:1)”, “View 1 (2:1)”, etc.

View state (current, background, or hidden; see Section 3.5.4.2 on p. 1162 for details) is displayed in the Drawing Tree as “c”, “b”, or “h” letter in parentheses to the right of the view icon (Fig. 3.5.8).



The Drawing Tree also displays macro elements, view inserts and fragment inserts contained in the drawing. See Section 3.7 on p. 1237 for details.

Icon size in the Drawing Tree can be customized, see Section 9.1.3.8.7 on p. 1690.

You can manage displaying some object types in the Tree using the **Drawing tree composition** button in the upper part of the window.

If the Drawing Tree elements are highlighted (models and views) the respective objects are highlighted in the drawing window.

Using the Drawing Tree elements' context menu, you can manage the status and some parameters of the image. For example, commands of the view's context menu (Fig. 3.5.9, a) can be used to explode or delete a view; commands of the fragment insertion context menu (Fig. 3.5.9, b) can be used for the insertion editing.

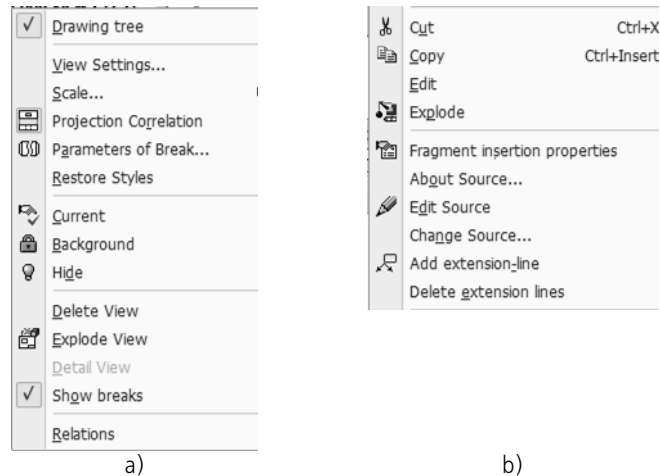


Fig. 3.5.9. Context menu of Drawing Tree elements:
a) view (associative), b) fragment insertion

3.5.4.6. View Caption

The caption is a text object that is part of the view contents.

The view caption is used to automatically create and update the text accompanying:

- ▼ a cut,
- ▼ Section view
- ▼ Enlarged Detail,
- ▼ a view in arrow direction,
- ▼ a node point designation,
- ▼ a sectional node point designation.

A view caption consists of the elements (designation, “reoriented” mark, scale, etc.) any of which can be included into or excluded from the view caption at the user's discretion. To do this, use the interface items on the **View Caption** tab that appears on the Property bar when creating or editing a view (see Section 3.5.4.6.3).

Each element of a view caption is associatively related with a layout object or a parameter of the view itself (see Section 3.5.4.6.1, 3.5.4.6.2).

The presence of designation symbols for cut lines, enlarged details, view arrows and node designations implies the presence of the respective images in this drawing. Therefore, to speed up expedite working, once any of the listed designation symbols is created, the view creation command is launched automatically. The caption of this view is associatively related to a newly cre-

ated layout object. The user just needs to set up, if necessary, other view parameters (including its caption) and specify position of the view on the drawing. After that, you one can proceed to drawing the image in the view.

The view caption can be edited:

- ▼ as part of a view — when creating or editing the view, see Section 3.5.4.6.3 (for non-associative views) and 3.6.1.3.4 on p. 1214 (for associative views),
- ▼ as text object (see Section 3.5.4.6.4)

3.5.4.6.1. Layout Object Linked to a View

The layout object linked to a view — designation of an enlarged detail, cut line, view arrow, node or sectional node which serves as the source of letter or numerical designation in the view caption.

A relation between a view and a layout object can be established in two ways.

- ▼ **Automatic.** The view that is automatically created after performing the command of placing a layout object (**Cut Line, Enlarged Detail, View Arrow, Node Designation, Sectional Node Designation**), will be linked to the object created by this command. Associated Views¹ **Cut/Section, Enlarged Detail, View in Arrow Direction** are linked with the layout objects selected to construct them.
- ▼ **Manually.** A link is manually created in the view caption to the required layout object. A link can be created using the **Link** button on the **Caption** tab of the Property bar while creating (editing) the view or while editing the view caption as a text object. This method is available only for non-associative views.

Due to such relation, layout object symbols and view captions always match each other.

3.5.4.6.2. Associative relation between the view caption and layout object designation

View caption consists of the following elements:

- ▼ Letter or numerical symbols
- ▼ “Unfolded” mark
- ▼ View scale
- ▼ “Rotated” mark
- ▼ Rotation angle
- ▼ Sheet number
- ▼ Zone name.



The caption of a view, where a detail or detail section are drawn, can contain only digital symbols and the page number.

A layout object symbol consists of the following elements:

- ▼ letter or number,

1. Associative views are detailed in Section 3.6 on p. 1205.

- ▼ Sheet number
- ▼ zone number.

All elements of a view caption and layout object designation (except for a letter or number of the latter¹) are links. The link sources for a view caption are the layout object linked to the view and the properties of the view itself. The link source for a layout object symbol is the view, to which this symbol is linked. Links produce associative relations between a view caption and a designation symbol of a layout object linked to it. Due to such relations, a view caption and layout object designation always match each other. The user can include or exclude any element at any time and therefore enable or disable any link.

All possible links and their direction are schematically shown by arrows in Fig. 3.5.10. Arrows point from the source to the link.

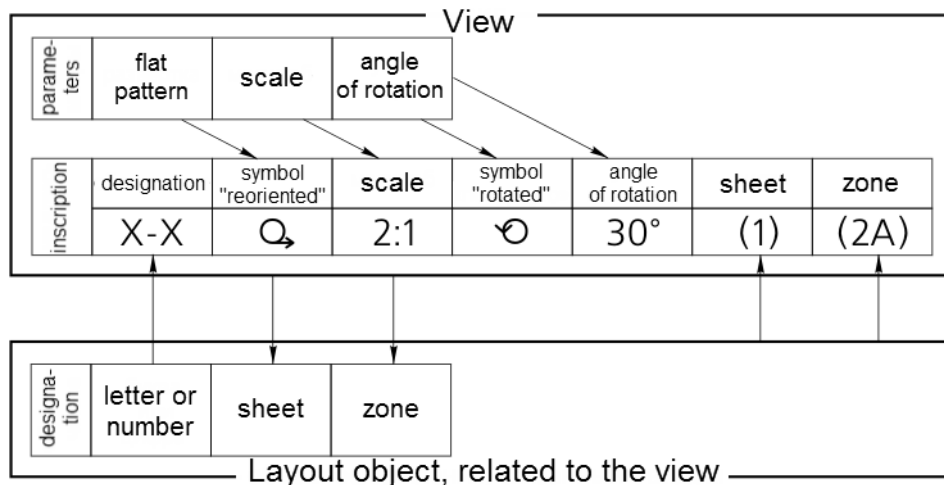


Fig. 3.5.10. The diagram of links between the view caption and the cut line designation (for mechanical engineering) related to this view

Consider an example of using the links between a view caption and a layout object designation. A view is created in a drawing that contains an enlarged detail. Enlarged Detail designation: *B (1A)*, view caption: *B (2:1) (2C)*.

The following changes were introduced in the drawing later:

- ▼ several more images were added, resulting in the enlarged detail designation symbol changing to *D*,
- ▼ The scale of the view containing the enlarged detail is changed: the new scale is 2.5:1.
- ▼ the view is moved to the zone 2A.

After those changes, the enlarged detail designation symbol automatically turns to *D (2A)*, while the caption of the view containing the enlarged detail becomes— *D (2.5:1) (2C)*.

1. A letter or number of the layout object designation is detected by the system automatically and entered by the user manually.

3.5.4.6.3. Editing View Caption as a Part of a View

A view caption is set up using the controls of the **View Caption** tab on the Property bar (Figure 3.5.11).

These elements are described in Table 3.5.4.

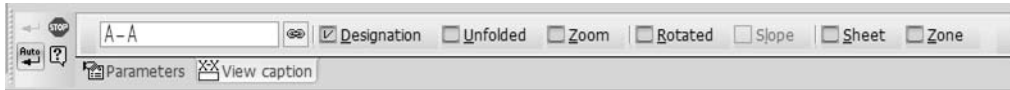


Fig. 3.5.11. **View Caption** tab

Tab. 3.5.4. View caption controls

Element	Description
View Caption Preview Window	The reference field containing the current view caption.
Paste Link	The button allowing to create or modify the link to a layout object in the view caption. After clicking the button, the Create Link dialog will appear. (Figure 4.1.28 on p. 1338). Select the desired source type, define the rest of link parameters and click the OK button. Once the link is created, an associative relation is formed between the view and the layout object selected as the link source. The link text will appear in the View Caption Preview Window of the Property bar. The options Designation , Sheet and Zone will become available.
Designation	Disabling this option breaks the link of the view with the layout object; therefore, enabling the option later becomes impossible. To create a link anew, click the Paste Link button.
Reoriented	If the option is disabled, the “unfolded” icon is not included in the view caption. If the option is enabled, the icon is included.
Scale	This option allowing to include the current scale in the view caption.
Rotated	If the option is disabled, the “rotated” icon is not included in the view caption. If the option is enabled, the icon is included.
Angle	The option allowing to include the rotation angle value in the view caption. The Angle option is available if the Rotated option is enabled.
Sheet	The option allowing to include the link to the drawing sheet number, where the layout object associated with the view is located, in the view caption. The Sheet option is available if the Designation option is enabled.

Tab. 3.5.4. View caption controls

Element	Description
Area	<p>The option allowing to include a link to the drawing zone designation symbol, where the layout object associated with the view is located, in the view caption. If the option is enabled, the zone designation symbol will be added to the caption provided that zoning is enabled in the drawing. If the option is disabled, then the zone designation symbol is not added to the view caption.</p> <p>The Zone option is available if the Designation option is enabled.</p>

3.5.4.6.4. Editing a View Caption as a Text Object

Since the view caption is a text object, you can use the techniques typical for such objects to edit the view caption.

To change position of the view caption, select it and then move and/or rotate with the mouse. You can also use the **Edit Location** command. It is available, if the view caption contains normal text (not a link).

Please note that, if a view is rotated, then the view caption rotation angle will change to compensate that in order to maintain the symbol being parallel to the X-axis of the global absolute coordinate system.

To modify the contents and/or the layout of the caption, double-click it. The text editing process will start. Tabs with interface items to set up the view caption will appear on the Property bar:

- ▼ The **View caption** tab makes it possible to change the caption's contents; for description of the tab's controls, see Table 3.5.4 (the only difference is that when editing a view caption as a text object, no preview window on the tab),
- ▼ the **Format** tab serves to edit caption text formatting (e.g. select a font, set up paragraph parameters, etc.),
- ▼ the **Insert** tab serves to insert various objects into a caption, for example, text templates, special characters, symbols, etc.

The following actions can also be performed in the editing mode:

- ▼ entering an arbitrary text anywhere in the caption,
- ▼ modifying parameters of the links inserted in the caption by the context menu command **Edit Link** (Fig.3.5.12),
- ▼ convert links to normal text by the context menu command **Explode Link** (Fig.3.5.12).

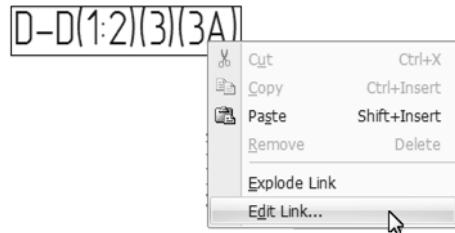


Fig. 3.5.12. Editing and breaking links in view caption



You cannot modify the links to a layout object in an associative view caption. Such modification would mean breaking the relation between the view and the layout object and would make the existence of an associative view impossible.

You can copy and move view captions using the clipboard. After the view caption is inserted from clipboard, it becomes text, i. e., it stops being a property of a specific view. In this case, if the caption is inserted in the same document from which it was copied, then the links existing in the caption still maintain their relations with the sources. If, on the other hand, a view caption is inserted in another document, then the relations of links with their sources are broken—the links are displayed in red.

3.5.5. General View Handling Techniques

For your convenience, KOMPAS-3D provides various ways of using views. Almost every action can be carried out in several different ways.

For example, commands can be selected from context menus of the views in the Drawing Tree. Enabling and using the Drawing tree is described in detail in Section 3.5.4.5 on p. 1163.

The **Document manager** provides convenient ways to control view states of a drawing. To open it, do one of the following:



- ▼ Call command **Service— Document manager**.



- ▼ click the **View Management** button in the **Current Status** tab



- ▼ click the **Layer Management** button in the **Current Status** tab

- ▼ select **Insert— Layer...**

A detailed description of the **Document manager** interface and ways to work with it is provided in Section 3.5.6.2 on p. 1182. This section describes the ways of working with views using only the **Document manager**.

3.5.5.1. Switching between Views

A view can be made current, and the states of other views may be set up using the **Document manager** (see Section 3.5.5.2).

However, there are faster ways to select a new current view.

- ▼ Select or type in the number or name (depending on the setup, see Section 9.1.7.8 on p. 1710) of a required view in the **Current view** field in the **Current state** panel (Figure 3.5.13), to make it current. The view selected in from the list, is highlighted in the document window. Icons next to the number or name of the view indicate its current state and color.

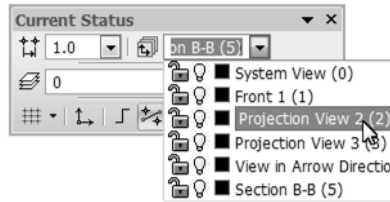


Fig. 3.5.13. Selecting the current view

- ▼ Select the desired view in the drawing's Drawing Tree and call the context menu command **Current**.
- ▼ If the required view is enabled, i. e., is active and visible, double-click any object of this view. The object editing process will be launched and the view, in which the object is located, will become current. The view that used to be current will return to its previous status.

3.5.5.2. Changing the view state

To change view status, complete the following steps.

1. Call the **Document manager**.
2. Select the root object in the Tree of Sheets, Views and Layers of Manager, which is the current drawing. The List of Sheets, Views and Layers will display all views existing in the drawing and their properties.
3. In the **List**, select the view which state needs to be modified.
4. Set the view properties using the buttons on the **Document Manager** toolbar:



- ▼ **Make current**



- ▼ **Active**



- ▼ **Background**



- ▼ **Visible**

**▼ Hidden**

You can alter the states of several views in a similar manner. To do this, select them and click the necessary button on the toolbar of the **Document Manager**.



To make all views of a drawing or all layers of a view (fragment) active and visible, click the button **Enable All** on the toolbar of the **Document Manager**.

The values of view properties can also be modified by clicking the icons in the fields of the **List of Sheets, Views and Layers**.

5. Close the dialog **Document Manager** by clicking the **OK** button.



Sometimes, it might be easier to use other methods to change the view state:

- ▼ the commands **Current**, **Background**, **Hide** from the view context menu in the Drawing Tree,
- ▼ using the icon for **Activity** and **Visibility** properties in the view list in the **Current state** panel (Figure 3.5.13); clicking an icon reverses the property.



Remember that you cannot change properties of the current view.

3.5.5.3. Changing the Properties of a View



In order to change parameters of the current view, click the **Current view parameters** button in the **Views** toolbar or call the respective command from the **Service** menu.

If the Drawing Tree is visible, you can edit the properties of any active view of the drawing, not just the current view. Select **View properties** in its context menu.



The elements shown in Table 3.5.3 on p. 1163 will appear on the Property bar.

Set the properties of the view as required and click the **Create object** button.



If only the scale has to be changed, use the **Scale** command in the context menu of this view in the Drawing tree.

If the scale of a view containing extension lines and leaders was changed, a request for scaling these objects is displayed. You can confirm or deny scaling (for details, see Section 3.4.4.2.1 on p. 1131).

You may change the properties of any kind of drawing view except the *System* one. This view has the following properties:

- ▼ Name *System view*
- ▼ Number *0*
- ▼ Color *Black*
- ▼ Scale *1:1*

- ▼ Anchor point $0,0$
- ▼ Rotation angle 0°

3.5.5.4. Selecting a View

One or more views can be selected for subsequent editing, copying to clipboard, deleting, etc. You can select only the current and active views.

Around the selected objects a rectangular highlighted outline frame is displayed. It indicates that the view is selected as a whole (part of the drawing), not just the objects in this view.



If the view is empty, then upon its selection a colored square with its center in the origin of the view coordinate system will be displayed.

To select a view, call the **Select – View** command in the main menu. A submenu will open, containing the commands described in Table 3.5.5.

Tab. 3.5.5. The commands for views selection

Command	Description
By Clicking	Selects one or more views by spotting an object belonging to these views.
From List	Selects views by name. Upon command activation, a dialog box opens containing a list of views available in the current drawing. Select required view in the list and click OK .

A view can be selected by clicking its icon in the Drawing Tree, as well.

3.5.5.5. Copying and Moving Views via Clipboard

An existing view can be copied or moved to another drawing with all its objects. For this purpose, the KOMPAS-3D clipboard is used.

To copy or move a view, select it and place it into the clipboard. Then paste it into another drawing. For details about working with the clipboard, see Section 3.1.3.5 on p. 882.

If a view with such ID is already present in the drawing, the system checks whether the view contents in the clipboard can be inserted into it (combine views). The following properties must match to enable views merging:

- ▼ Numbers
- ▼ Base point coordinates
- ▼ rotation angles
- ▼ Scales¹.

If merging is impossible, a new view is created and given the next available number.

1. System views always fit these conditions.

If a view is to be copied into the same drawing, it is more convenient to use the mouse (for details, see Section 3.4.1.1.2 on p. 1114). The system creates a new view and gives it the next available number.



If necessary, you can paste an image from another drawing's view by creating a special object—a view insert (see Section 3.7.3 on p. 1251).

3.5.5.6. Deleting a View

If the contents of a view is no more needed, the view can be deleted from the drawing.

To delete a view, select it and then select **Edit — Delete— Selected objects** in the main menu, or press <Delete> on the keyboard.

Alternatively, select **Delete** in the context menu of the view in the Drawing Tree.

The Delete Views dialog will appear. It lists the names of the views to be deleted. Confirm the deletion by clicking **OK**, or cancel the deletion by clicking **Cancel**.



The system view cannot be deleted. You may delete only the objects contained in it (see Section 3.4.8 on p. 1144), while the view itself is always present in the drawing.

Views can be deleted in **Document manager** as well. To do this, perform the following actions.

1. Call the **Document manager**.
2. Select the root object in the Tree of Sheets, Views and Layers of Manager, which is the current drawing.
The List of Sheets, Views and Layers will display all views existing in the drawing and their properties.
3. In the **List** select the view (views) to delete.
4. Click the **Delete** button on the **Document Manager** toolbar.



Current and system views cannot be deleted from the **Document manager**.

If the deleted view (views) contains objects, a notifying message will be displayed in this regard. An icon of the view in the **Document manager** indicates whether it contains objects or not:



▼ Empty View,



▼ A view with objects.

5. Confirm the deletion.
6. Close the dialog **Document Manager** by clicking the **OK** button.

3.5.5.7. Placing the Views on a Sheet

When placing the views on a drawing, their scale and position in a sheet can be changed.

The drawing arrangement becomes much simpler if each image is placed in a separate view. In this case, the scale, position and the rotation angle of any image can be quickly changed.



Placement actions can be performed only with the current and active views of a drawing.

To change the scale of a view, call its context menu in the drawing's Tree and select the desired scale from it (Fig.3.5.14).

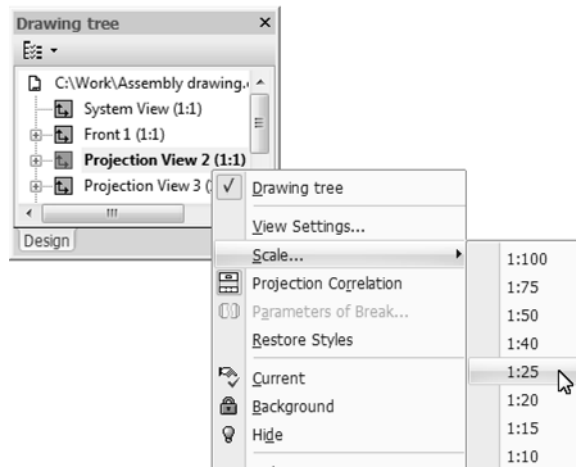


Fig. 3.5.14. Changing the scale of a view

After that, the geometrical objects (line segments, arcs, etc.) contained in the view will update—grow or shrink according to the specified scale. At the same time, the lines width, arrows length and font size in annotation labels, symbols size etc. will not change. The dimension values will also remain the same. For example, in Fig. 3.5.15, a, the scale of the view that contains a plug is 2:1. To fit specifications on the sheet, the image needs to be shrunk. To do this, change the view scale to 1:1 (Fig.3.5.15, b).

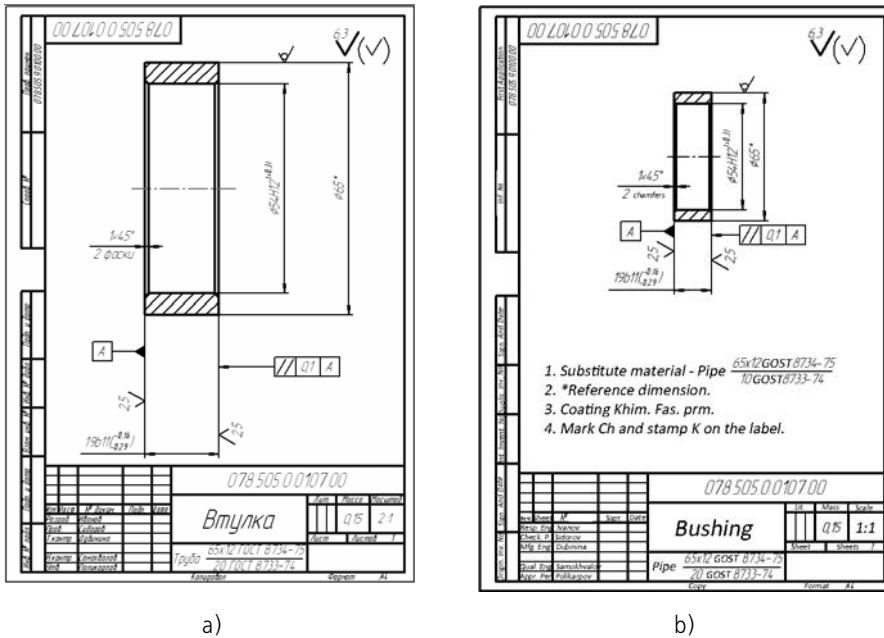


Fig. 3.5.15. Modifying a view scale

To change the position of a view, select it (see Section 3.5.5.4 on p. 1173), and move it to a new place using the mouse. Set the cursor inside the outline frame of the view hold left mouse button and drag the mouse. The view will move after the cursor. When the desired view location is reached, release the mouse button.

To change the rotation angle of a view, call the **View Settings...** command from its context menu in the Drawing Tree. The Property Bar will display controls for setting up selected view's parameters. Enter the rotation angle value in the **Angle** field on the **Options** tab of the Property bar. View scale and anchor point coordinates can be set in the same tab, if necessary.



The system view settings cannot be edited.



Some associated views are linked to their reference view. In this case, the link should be disabled before rotating or moving the view.

The following editing commands are applicable to views:

- ▼ **Shift** (see Section 3.4.2 on p. 1120),
- ▼ **Rotation** (see Section 3.4.4.1 on p. 1129),
- ▼ **Scaling** (see Section 3.4.4.2 on p. 1130),
- ▼ **Symmetry** (see Section 3.4.4.3 on p. 1131).



Command execution results in **scaling** the whole view, not just the objects; command execution results in creation of a new view in the drawing, **symmetrical** to the selected one.

3.5.5.8. Break in view

According to the standard, long items or their elements with constant or regularly changing sections may be drawn with breaks, i. e. with their middle part conditionally removed. An image may contain one or several breaks.

The **Break in View** command is used for such image transformation in KOMPAS-3D. This allows to conditionally remove a specified portion (portions) of the image and move the remaining portions closer together. The command can be applied equally to associative and to non-associative drawing views.

The system supports several types of break lines — **Bent**, Straight and so on. The default setup of the break line – selection of the break line type and settings of its parameters – is described in Section 9.2.6.7 on p. 1766,



Breaks in the images are not possible in a system view.

If you need to create breaks in a view, it is recommended to follow the working order below.

1. Creating all geometrical objects in a view.
2. Creating the required number of breaks in the view.
3. When working with associative views, creation of views that use the break view as the base reference one.
4. Adding layout objects to a view: dimensions, designations, annotations etc.

3.5.5.8.1. Creating a Break





To create an image break in the current view, select the **View Break** command.

Two parallel lines will appear on the screen, representing the boundaries of the break (Fig. 3.5.17 on p. 1179), while on the Property Bar the control items to set up the break parameters will become visible. These control elements are shown in the Table 3.5.6.

By moving the defining points of the break boundary with the mouse, surround the portion of the image to be deleted.

Tab. 3.5.6. Control elements to break parameters

Element	Description
Breaks	The bar containing the list of LCSs in the current document. The break selected in the list is considered to be the current one.
	The Add button serves to create a new break in the view. After clicking it, a new pair of break boundaries will appear on the screen.
	The Delete button is used to delete the current break from the view.

Tab. 3.5.6. Control elements to break parameters

Element	Description
Angle	The field for entering or selecting an angle between the X axis of the coordinate system in the current view and the shift direction (see Section 3.5.5.8.2) of the current break.
Gap*	This field serves to enter or select the distance between the break boundaries — the distance, at which the shown portions of the image will be after creating a break.
Line Type	List for selection of the break line type.
Amplitude, % Amplitude, max	Fields for setting the height of wavy or bent line. See Section 3.5.5.8.3 on p. 1180 for detailed description of these fields.
Show breaks	The option enables the current view displayed with breaks. When this option is switched off, the image in the view is displayed in full (breaks are preserved). Note that a view may be displayed either entirely, without any breaks, or with all breaks created in it. You cannot deactivate just one of the breaks.

* When using bent break lines, the gap should be not less then the doubled setback amount twice the height. The minimum gap when using lines of other types is 1mm.



When setting boundaries, the distance between the break boundaries (see Fig. 3.5.17 on p. 1179) cannot be less than the double value of amplitude.



To confirm the creation of the break, select the **Create Object** button.

The break will be created (Fig. 3.5.16). All geometrical objects of the current view that were inside the break boundaries will not be displayed on the screen. The visible portions of the image will be limited by the break lines and moved towards each other so the distance between them becomes equal to the value defined in the **Gap** field. The length for the lines of break is determined automatically by the system from the image outlines on the view.

In the context menu of the break view in the Drawing tree, commands to control the break become available: **Parameters of Break...** and **Show Breaks**. They are represented in Table 3.5.7.

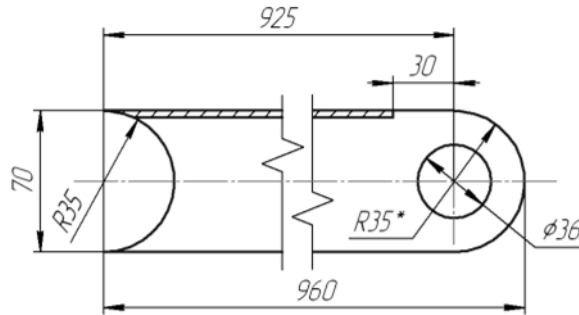


Fig. 3.5.16. Example of image break

Tab. 3.5.7. Commands to Control a Break

Command	Description
Parameters of break...	Allows to set up image breaks on a view. After calling the command, the Property bar displays the same controls, as when creating a break. You can change parameters of any breaks in a view or delete any break.
Show Breaks	Allows to control the image of a view: toggle the display of breaks*. Note that a view may be displayed either entirely, without any breaks, or with all breaks created in it. You cannot deactivate just one of the breaks.

* This command works similar to enabling (or disabling) the **Show breaks** option on the Properties bar when creating or editing breaks.

3.5.5.8.2. Shift Direction for a Break

Shift direction is the line along which the visible pieces of the image are moved towards each other when creating a break in a view. Figure 3.5.17 shows the view coordinate system, break borders and shift direction. In this case, the angle between the X-axis and the offset direction is zero.

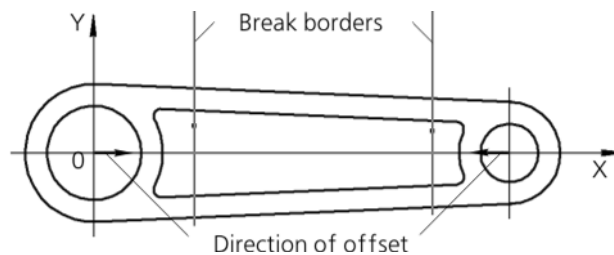


Fig. 3.5.17. Shift direction

Note that the directions of shifting the same view can be only parallel or perpendicular to each other. For example, a break is created in a view and its shift direction is at the 30 degree angle

to the X axis. Later in this view you will be able to create breaks with shift directions, which are either at the same angle or at the angle of 120 degrees to the X-axis.

3.5.5.8.3. Amplitude

Amplitude is deviation of a bend line or wave line from the centerline (Fig. 3.5.18).



Fig. 3.5.18. Height: a) of a bent line,
b) of a wavy line (the middle line is also shown in the figure)

The setback value for a bent break line is equal to the value defined in the **Height, max** field.

The height value for a wavy line is set in the **Amplitude, %** as a percentage of the **break length**.

Break length is the maximum distance between the outermost visible points, measured in the direction perpendicular to the shift (Fig. 3.5.19).

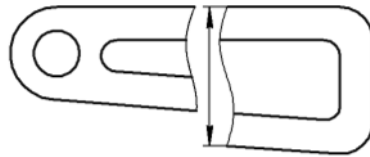


Fig. 3.5.19. Break length

The value entered in the **Height, max** field defines the maximum allowed setback for a waves line. If the calculated setback value in millimeters exceeds the maximum, then the maximum value, rather than the calculated one, will be used to draw the line.

3.5.5.8.4. Special Issues for Working with Image Breaks

1. If the grid display was on in the drawing window, then it will not be displayed between the boundaries of the break. Snapping to grid between the break borders is impossible.
2. All dimensions, designations, notations and tables existing in a broken view stay visible even if they were positioned (fully or partially) within the boundaries of the break when it was set up.
3. Centerlines and center marks obtained by using commands **Centerline by two points**, **Auto Centerline** (see Section 3.3.7.15 on p. 1049 and 3.3.7.16 on p. 1050) and **Center Mark** (see Section 3.3.7.14 on p. 1048), are not broken. But if the specified objects were created in a document using the **Segment** command (see Section 3.2.4.1 on p. 906), they will be broken just like the rest of geometrical objects.



4. Upon adding new objects to a broken view, the break lines are not rebuilt automatically. To re-draw them in according to the new outline dimensions of the image, select the **Refresh Image** command.
5. An image break is a feature of a drawing view. The view contains the information about the break lines location relative to the coordinate system of this view, but not relative to the objects in the view. Therefore, when you somehow move objects in a view, break lines remain in their original places. If required, you can edit a break by using the **Parameters of Break...** command. (see Table 3.5.7 on p. 1179). If you need to change the location of objects together with a break, modify a view location (see section 3.5.5.7 on p. 1174).

3.5.6. Layers

If in conventional paper-and-ink drafting you have experienced creating drawings or diagrams with a great number of elements (lines, dimensions, symbols, etc.), then you know about tracing paper being used for developing layouts of similar design documents. In this case, the designer places various graphical information blocks as if on different logical (in terms of selecting this block as a separate unit) and physical (in terms of placing the block on a separate paper medium) levels.

Such division makes it much easier to arrange image elements on the sheet and edit them. At each stage of document (drawing) development, only those the blocks are used that are required at the moment.

In KOMPAS-3D, this method can be realized with the help of layers. Each drawing view (see Section 3.5.4 on p. 1160) and each fragment can contain up to 2,147,483,647 layers.

Explicit division of a fragment or a view into layers is not mandatory for user. When a new drawing fragment or view is created, it automatically forms the layer number 0, which can be used right away.

To create new layers and manage them, use the **Document manager**. It is described in detail in Section 3.5.6.2 on p. 1182.

3.5.6.1. Layer States

The state of a layer is defined by the values of the following parameter values:

- ▼ **Active,**
- ▼ **Visibility,**
- ▼ **Print.**

The **Activity** property manages the availability of layer objects for editing, and can have two states: **active** or **background**.

Objects of the active layer are accessible for editing and deletion operations. All contents of an active layer are displayed in the same color, set for the current layer in the **Document Manager dialog**. Active layers can be compared to tracing paper, which is not being modified, but with only the previously drawn elements are corrected and the tracing papers themselves are moved around on the a conventional drawing board.

A background layer is accessible only for the operations of snapping to points or objects. Such a layer cannot be moved, and its objects are not accessible for editing. The contents of all elements of background layers are displayed on the screen in the same style that can be customized (see Section 9.1.7.9 on p. 1711). Background layers are similar to tracing paper, fixed to the drawing board for lining on them the new elements of a drawing or scheme.

The **Visibility** property manages the display of a view and also has two states: **visible** and **hidden**.

If a layer is visible, its objects are displayed on the screen. In this case, the active layers are shown in the colors selected for them, while the background ones are shown in the specified style.

If a layer is hidden, then it is not displayed on the screen regardless of whether it is active or background. Therefore, a hidden layer is completely inaccessible for any operations. Comparatively speaking, you can remove the currently unnecessary tracing papers from a conventional drawing board.

The **print** property controls the layer's printing options and may have the following values: **allow printing** and **suppress printing**.

Among all view or fragment layers, there is one and only one that has the **current** status.

The current layer of the current view is the place where all the newly created objects are stored.

Any layer can be made current. In this case it automatically becomes visible and active. While the layer remains current, these properties cannot be changed (i. e., the current layer cannot be made hidden or put into background). At the same time, printing of the next layer can be suppressed. The **print** property value does not change if the **current** layer status is assigned or removed.

Objects of the current layer are displayed in the real styles of lines, points and hatches, which are set in the system settings dialogs. It is possible to compare the current layer with the tracing paper, which is located on top of all others and on which the engineer is currently drawing on.

Once the **current** status is assigned to another layer, the state of the layer that used to be the current before is restored. For example, at some moment of working with the drawing, Layer1 was current. The Layer 2 was both background and visible at the time. Then, the Layer 2 was made current, so it became active. After finishing editing Layer 2, Layer 1 becomes current again. Layer 2 became background again at the same time.



Depending on the system settings (see Section 9.1.5.2 on p. 1701), hidden layers can be printed or not.

3.5.6.2. Document Manager

The **Document Manager** serves to work with objects, composing the structure of the document: sheets, views and layers. Main actions performed with the **Document Manager** are:

- ▼ Changing properties of Sheets, Views and Layers
- ▼ Creating sheets and layers
- ▼ Deleting sheets, views and layers,
- ▼ Selecting current view and layer

- ▼ Grouping layers
- ▼ Copying layers from view to view.

The **Document Manager** can be called in a text document and BOM. It is used for viewing and changing symbols used in basic and auxiliary document sheets (for BOMs — style of main BOM sheets). You can also perform the following actions with respect to additional sheets: adding, deleting, changing the format, multiplicity or orientation.

In order to open the **Document Manager**, perform one of the following actions;



- ▼ call the **Service— Document manager...** command,
- ▼ select **Insert— Layer...**



- ▼ click the **Document manager** button on the **Standard** panel.
- ▼ click the **Layer Management** button in the **Current Status** bar,



- ▼ click the **View Management** button in the **Current Status** bar,
- The **Document manager** dialog appears on the screen (Fig. 3.5.20).

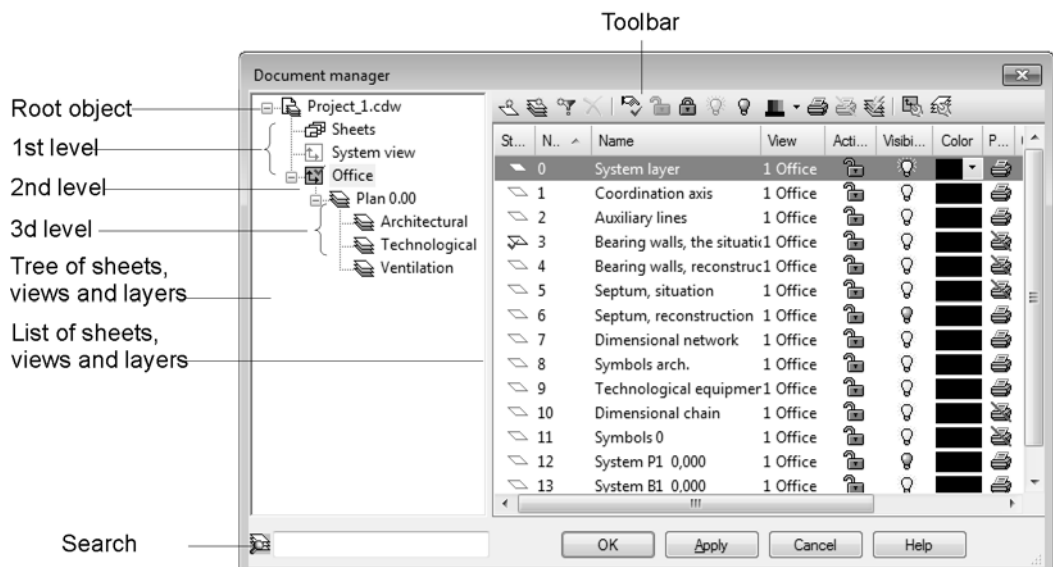


Fig. 3.5.20. **Document Manager**

The **Document Manager** dialog contains the following areas:














- ▼ Toolbar (see Section 3.5.6.2.1),
- ▼ Tree of Sheets, Views and Layers. (see Section 3.5.6.2.2),
- ▼ List of Sheets, Views and Layers. (see Section 3.5.6.2.3),
- ▼ Search area — only for graphic documents (see Section 3.5.6.2.4).

3.5.6.2.1. Toolbar of the Document Manager




The toolbar is located in the upper part of the **Document manager** window. The set of buttons on the toolbar is determined by the type of the document. For a graphics document, the set of buttons depends also on the current element selected on the tree of sheets, views and layers.

Controls for working with views and layers in a graphics document are described in Table 3.5.8.

Tab. 3.5.8. The **Document manager** toolbar for working with views and layers

The	Function
	<p>Create Layer Create new layer. The command is inaccessible in the following cases:</p> <ul style="list-style-type: none"> ▼ Several objects are selected; ▼ Document, filter or group on the first level is the current object; ▼ Current is the object from the List of Sheets, Views and Layers
	<p>Create Group Create a group of layers. button is enabled if layer grouping is enabled for the document*.</p>
	<p>Create Property Group Create Property Group. The button is enabled if layer properties grouping* is enabled for the document.</p>
	<p>Create Filter Create new layer filter. The button is enabled if layer grouping is enabled for the document*.</p>
	<p>Delete Serves to delete an object or selected objects. When attempting to delete an empty view or layer, a warning message is generated. You can confirm deleting the view (layer) with all its objects, or cancel the deletion.</p>
	<p>Make current Make current the selected layer.</p>
	<p>Active Serves to make a view, layer, selected layers or all layers in a group active, i.e., that is accessible for editing.</p>
	<p>Background Serves to make a view, layer, selected layers or all layers in a group background, i.e., that is inaccessible for editing.</p>
	<p>Visible Make visible a layer or view, the selected layers, or all layers in a group.</p>
	<p>Hidden Make invisible a layer or view, the selected layers, or all layers in a group.</p>
	<p>Color Set the color for a layer or a group. After the color is set, the button will be shown in this color.</p>
	<p>Enable printing Allow output of selected layers for printing. The command is unavailable, if layer printing is already allowed for all selected objects.</p>
	<p>Suppress printing Suppress output of selected layers for printing. The command is unavailable, if layer printing is already suppressed for all selected objects.</p>






Tab. 3.5.8. The **Document manager** toolbar for working with views and layers

The	Function
	Enable All Make all hidden and background layers of the view, selected in the Tree of sheets, views and layers, (or all views of the current drawing) active and visible. The button is disabled if there is are no selected objects in background or hidden state.
	View Settings Open a dialog box for setting up the display of the current document views.
	Layer Settings Open a dialog box for setting up the display of the current document layers.




* Layer grouping mode can be selected while grouping setup, see Section 9.2.6.3 on p. 1760.

Controls for working with views and layers in a graphics document are described in Table 3.5.9.

Tab. 3.5.9. **Document Manager** toolbar

The	Function
	Create Sheet <ul style="list-style-type: none"> ▼ Create a new sheet in a drawing. The button is available if Sheets is the current object. ▼ Create a new sheet in BOM. It is available, if the current object is Additional sheets in the beginning/end of the document.
	Create section Create a new section in a text document. Available in a text document.
	Format Set the format of the current sheet or section. Available if the sheet or section is current, while BOM has an additional sheet.
	Layout Serves to set the template library and the layout style from this library. It is available, if a sheet or section is the current one (document sheet, additional sheet).
	Move Down/Up Changing the order of additional sheets in the beginning or at the end of BOM. Order of additional sheet sequence in a document is defined by sequence of allocation of their layout in dialog lists. Once the button has been clicked, the additional sheet is moved one position up or down. The button is available, if the sheet is not the first or the last one among the sheets in the Additional sheets in the beginning/end of a document list .

Tab. 3.5.9. **Document Manager** toolbar

	The	Function
	Delete	Delete additional sheet in the graphics document, section in a text document, or additional sheet of BOM. It is available, if the current sheet is an additional one or an object in the Additional sheets in the beginning/end of the document . Once the object is specified, all sheets are deleted from it.
	BOM Settings	Calling a setup dialog for general style and section styles of the current BOM.
	Settings...	Bring up the current specification parameter dialog.

3.5.6.2.2. Tree of sheets, views and layers

Tree of Sheets, Views and Layers is located on the left side of the **Document manager** window.

The **Document** is a root object of the Tree.



▼ fragment or drawing,



▼ text document,



▼ BOM.

The root object's name is the same as the file's name. The **Document** node can may contain the following first level subordinates:



▼ sheets (for a drawing), additional sheets and document sheets (for a text document and BOM),



▼ views (several views are possible for a drawing; only one is possible for a fragment):



▼ empty view,



▼ view, which contains objects,



▼ groups of layers (for a drawing or a fragment),



▼ groups of layers' properties (for a drawing or a fragment),



▼ filters (for a drawing or a fragment).

The views in the Tree can have the following subordinate objects:

▼ Layer groups,

▼ Layer Properties groups,

▼ Filters,

▼ View Inserts.

The groups in the Tree can have the subordinate groups from next levels. The number of nesting levels is unlimited.

Sheets, filters and view inserts do not have any subordinates.

An object selected in the Sheets, views and layers Tree is considered **current**.

The position of filters and groups in the Tree can be changed. This is done using the Drag & Drop method. When a group is moved to a new location, it is deleted from the old location. To place a group onto a new location without deleting it from the old location (i.e., to copy a group), click and hold the <Ctrl> key while dragging the group. When a group is copied, all its layers are also copied (for details, see Section 3.5.6.6 on p. 1191).











If you attempt to move the object to an invalid location, a warning sign will appear instead of the cursor.

3.5.6.2.3. List of sheets, views and layers

The list of sheets, views and layers can be found in the right part of the **Document manager** dialog. It contains the names of objects in the current element of the Tree. If the contents of the group include layers and groups, then in the List of Sheets, Views and Layers only the subordinate layers of this group are shown. The layers of the nested group are not displayed. To display them, make current the corresponding group current.

The columns of the Sheets, views and layers list show the properties as icons:

- ▼ **State:**
 -  ▼ empty view,
 -  ▼ view, which contains objects,
- ▼ **Activity:**
 -  ▼ active layer or view,
 -  ▼ background layer or view,
- ▼ **Visibility:**
 -  ▼ visible layer or view,
 -  ▼ hidden layer or view.
- ▼ **Print:**
 -  ▼ layer printing is allowed,
 -  ▼ layer printing is suppressed,

The current layer or view is additionally marked with a red check mark in the **Status** field.

Empty layer icon has a different contour color than that of a layer containing objects: it is gray in the former and blue in the latter.

Besides viewing the object properties, the Sheets, views and layers List allows editing the properties. Clicking a property value in the following fields changes it to its opposite value: **Status**, **Activity**, **Visibility**, **Print**. Clicking the **Number** or **Name** fields allows modifying the values of these fields. The object name can be changed by selecting **Rename** in its context menu. Clicking in the **Color** field opens the panel for choosing layer or view color.



You cannot rename a layer transferred from a model. The names of such layers start with a prefix 'c'. For more details about transferring layers from a model, see Section 3.6.3.10 on p. 1235.



Properties *Background* or *Hidden* cannot be set for *current* layer or view.

Objects, located in the List, can be sorted by any of the columns. To select a column for sorting, click on its name. A triangle will appear next to the name. It shows the direction of sorting. If the vertex of the triangle is faced down, then the sorting of the list is done by descending the values in this column, and if up— by ascending.



The **Sheets, views and layers list** also displays the views that are inserted into current drawing from other drawings, and the layers of these views. For special details of using these objects, see Section 3.7.1.7.2 on p. 1247.

3.5.6.2.4. Search Area



The **Search range** field is located under the Sheets, views and layers tree in the Document Manager of a graphic document. This field provide simplified filtering of layers by the **Number** and **Name** fields.

A simplified filter is applied to the elements of the object, which is current in the Tree of Sheets, Views and Layers, such as a document, view or a group. In order to begin the search, enter a text line into the field. In this case, you can use masks. The “*” character used in masks replaces any arbitrary number of any characters. “?” character replaces any single character. The List of Sheets, Views and Layers will display the objects, whose numbers and layer names are the same as the preset line. The contents of the search area are not saved in the document.

3.5.6.3. Creating New Layer

To create a new layer in the current view of a drawing or in a fragment, do the following.



1. Call the command **Insert — Layer** or click the **Layer States** button on the **Layer Management** bar. The **Document Manager** dialog will be displayed.



2. In the Sheets, Views and Layers List, select an object (view, group of properties, group of layers) to add a new layer and click the **Create layer** button in the **Document manager** toolbar.

The List of Sheets, Views and Layers will display the newly created layer. The text cursor will appear in the group name field. You can change the default name. Entering a name is not mandatory, but understandable logical layer names, especially if they are numerous, significantly facilitate the working with the document. For convenience, it is beneficial to add a commentary for a layer.



3. To set the layer display color in the active state, click the **Color** button on the toolbar of the **Document manager** toolbar. Layer color can also be selected from the drop-down list of the **Color** column.

Note that after creating a new layer, the current layer still remains the same as before. If you need the new layer to be current, click the **Make Current** button or click the mouse on the **Status** cell. On the layer icon, a red “check mark” will appear, indicating that the layer is current.

4. After finishing entering all new layer parameters, click the **OK** button of the Document Manager.

3.5.6.4. Switching between the Layers

You can make a required layer current, or set the state of other layers using the **Document manager** (see Section 3.5.6.2 on p. 1182).

However, there are faster ways to select a new current layer.

- ▼ Select or type in the number or name (depending on the setup, see Section 9.1.7.9 on p. 1711) of a required layer in the **Current layer** field in the **Current state** panel (Figure 3.5.21), to make it current. The view selected in from the list, is highlighted in the document window. The icons next to the layer number and name indicate its current state and color.

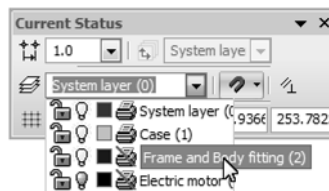


Fig. 3.5.21. Selecting current layer



The **Current Layer** field is displayed on the Bar if it is positioned horizontally.

- ▼ If the required layer is enabled, i. e., active and visible, then to make it current, do the following:
 1. Select any object, belonging to the desired layer.
 2. Call up the context menu and select the **Switch to Object Layer** command from it. The layer containing the selected object will become current.

The layer that used to be current will return to its previous state.



The **Switch to an object layer** command is disabled, if a “multi-layered” object (macro element, group, view or fragment insert) is selected, or if the selected object is already on the current layer.

3.5.6.5. Changing the Layer State and Properties

To change parameters and/or state of an existing layer, do the following.



1. Call the command **Insert — Layer** or click the **Layer States** button on the **Layer Management** bar. The **Document Manager** dialog will be displayed.
2. In the Tree of Sheets, Views and Layers, select an object (document, view, group of properties or group of layers), containing the layer, whose parameters need to be modified. The List of Sheets, Views and Layers will display all layers of the selected object.
3. Select the desired layer among them.



If you know the name or number of a layer, it can be selected using the Search area located in the lower left corner of the Document manager.

4. Set the state of the selected layer using the buttons on the Document Manager toolbar:



▼ **Make current,**



▼ **Active,**



▼ **Background,**



▼ **Visible,**



▼ **Hidden,**



▼ **Printing allowed,**



▼ **Printing disabled.**



You can alter the states of several layers at the same time in a similar manner. To do this, select them and then click the necessary button on the toolbar of the Document Manager or call the command from the contextual menu.



To make all layers of a view (fragment) active and visible, click the **Enable all** button in the Document manager toolbar.

The states of layers can also be modified by clicking the icons in the fields of the List of Sheets, Views and Layers.

5. If necessary, you can change parameters of the layer. number, layer name, color in active state, commentary.



You cannot rename a layer transferred from a model. The names of such layers start with a prefix 'c'. For more details about transferring layers from a model, see Section 3.6.3.10 on p. 1235.

6. After finishing editing state and parameters of a layer, click the **OK** button of the Document manager.

To quickly change the states of several layers simultaneously, layer sets can be used (see Section 3.5.6.9.1 on p. 1192).



It is convenient sometimes to change layers states using the **Activity**, **Visibility** and **Print** icons in the layers list in the **Current state** panel (Fig.3.5.21). Clicking an icon reverses the layer property that it represents.



Remember that you cannot change the activity and visibility of the current layer.

3.5.6.6. Copying Layers to Other Views

Layers can be copied from one view to another. While copying, the contents of source layers is ignored, i. e., the new (copied) layers are empty.

To copy layers, do the following.



1. Call the command **Insert – Layer** or click the **Layer States** button on the **Layer Management** bar. The **Document Manager** dialog will be displayed.
2. In the Sheets, Views and Layers List, select an object (view, group of properties, group of layers) containing the source layers, i.e. the layers to be copied. The List of Sheets, Views and Layers will display all layers of the selected object.
3. Select the layers to be copied and drag them onto the icon of the target view. If the copied layers must also be added to a group belonging to a view, drag the source layers onto the icon of the group.

Groups of layers and groups of layer properties including the constituent layers can also be copied between the views. To achieve this, drag the groups in the Sheets, views and layers Tree while holding the *<Ctrl>* key.

4. Close the Document Manager by clicking the **OK** button.



If copying a layer or a group to the specified location is allowed, a “plus” sign appears next to the cursor while dragging. If copying is not allowed, regular cursor is replaced by a warning symbol. For example, a group cannot be copied to itself or to the parent group. A layer cannot be copied to the view that contains it.



Layer numbering is verified during the copying. If the target view contains layers with the same numbers as the source layers, a warning dialog box opens informing the user about matching layer numbers (Figure 3.5.22). To continue, select one of the action buttons.

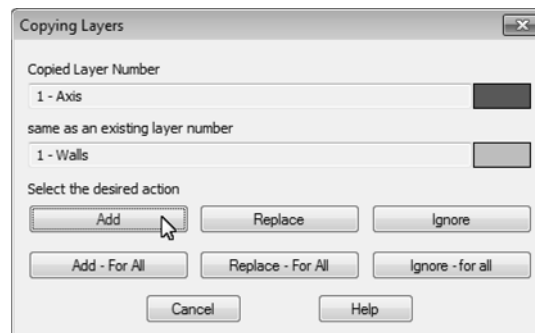


Fig. 3.5.22. Warning about Coincidence of Layer Numbers



A view can contain no more than only one system layer (number 0). Therefore, when copying a system layer from one type to another, **Add** and **Add— For All** buttons are not available.

3.5.6.7. Selecting a Layer

To select a layer, select **Select— Layer** in the main menu. A submenu will open containing the commands described in Table 3.5.10.

Tab. 3.5.10. Selecting a layer

Command	Description
By Clicking	Layers are selected by pointing on an object that belongs to the required layer.
From List	The layer is selected by entering its name. A dialog box opens containing a list of layers in the current fragment or view of the drawing. Select required layers in the list and click OK .

3.5.6.8. Deleting a Layer

You can delete a layer from the document. To do this, perform the following actions.



1. Call the command **Insert — Layer** or click the **Layer States** button on the **Layer Management** bar. The **Document Manager** dialog will be displayed.
2. In the Tree of sheets, views and layers, select an object (document, view, group of properties or group of layers) containing the layer to delete.



3. The List of Sheets, Views and Layers will display all layers of the selected object. Select the desired layer among them.

If you know the name or number of a layer, it can be selected using the Search area located in the lower left corner of the Document manager.



4. Click the **Delete** button on the Document Manager toolbar.



You can delete neither the current nor a system layer.

If the deleted layer contains objects, a warning message will be displayed in this regard.

5. Confirm the deletion.
6. Close the dialog Document Manager by clicking the **OK** button.

3.5.6.9. Managing Layers in a Graphic Document

If a document contains many layers, manual setup of each layer for various object groups might be inefficient. In such cases, it's much more reasonable to use **layer sets**. They allow to quickly change visibility, activity, ability to print and color of several (or all) layers of the document.

To create or work with a layer set, use the Document manager.

3.5.6.9.1. Layer Sets

Layer sets can be static, such as **Layer groups** and **Layer properties** sets, or dynamic, like **Filters**. Using a layer set in a document depends on the setup (see Section 9.2.6.3 on p. 1760).

Layer group

Using layer groups allows to simultaneously alter the activity, visibility and color of several layers in a same way. For example, you can make all layers in a group hidden.

A group may contain layers of a fragment, or layers of one or several views of a drawing. One layer can be included in several groups. Changing the activity, visibility, ability to print or color of a group, will change this option similarly for all layers, which belong to it. When a layer is included in a group, the actual properties of this layer are displayed in the group. The properties of the layer, which is included in the group, can be changed both in the contents of the group, as well as separately. If the properties are altered outside of the current group, then these changes will be displayed when viewing the group contents.

Adding a layer to the document does not make this layer automatically added to the group. Creating a layer in a group adds a layer to a document. The newly created layer is not added to other groups automatically.

When a layer is deleted from a group, it is not deleted from the document. Deleting a layer from a document makes it automatically deleted from a group (groups).

Groups are named objects. By default, when a group is created, it is assigned a name made up from the word Group and the sequential number. This name can be edited.

Groups of layers are saved in the document.

Layer Properties Group

A layer properties groups is a predefined **set of properties** applicable to the layers of the group.

A layer can be included in many groups. The value of the same property of the same layer may be different for different groups. The value of the same property of different layers in one group can be different, too as well.

When a group is applied, the properties of the layers included in the groups take assume the values specified in the group.

Below is an example of using a layer properties group. The following layer properties groups are created for View 1 of a floor plan.

Office Building (View 1)

- ▼ *Plan at the level 0.000 (Group 1)*
 - ▼ *Architectural designs (Group 1-1)*
 - ▼ *Engineering facilities (Group 1-2)*
 - ▼ *Ventilation and air conditioning (Group 1-3)*
- ▼ *Plan at the level 3.300 (Group 2)*
 - ▼ *Architectural designs (Group 2-1)*
 - ▼ *Engineering facilities (Group 2-2)*
 - ▼ *Ventilation and air conditioning (Group 2-3)*

Each group contains all layers of View 1.

Group 1-2 *Engineering facilities*, created for displaying the facilities located on level 0.000, has the following setup:

- ▼ layers containing architectural structures are background,

- ▼ the layers displaying ventilation and hosing systems will become hidden
- ▼ layers containing engineering facilities are active, one of them is current
- ▼ layers containing objects on level 3.300 are hidden

After Group 1-2 is applied, the layers in View 1 will change according to the values of this group:

- ▼ layers with ventilation and aspiration air conditioning system elements and layers with objects on level 3.300 will become hidden, i.e., invisible
- ▼ layers with architecture structures will become background, i.e., visible but uneditable
- ▼ layers with engineering facilities will become active, i.e., editable
- ▼ one of the layers with the facilities will become current, and newly created objects will be added to this layer

Group 1-1 makes active the layers containing architectural structures. The rest of the layers are hidden if they are not needed for working with these structures, and background if they must be visible.

Other layer properties groups are set up in a similar way.

Thus, using properties groups allows making prompt changes to the display and editing modes of objects in a graphical document according to current tasks, using the applying predefined sets of settings.

The layer properties will take the set values only after the group will become current (see *SectionLayer Properties Group* on p. 1198).

Since a group of layer properties is a combination of settings, therefore, the actual state of the layers included in its contents is not reflected in the group.

Adding a layer to the document does not make this layer automatically added to the group. Creating a layer in a group adds a layer to a document. The newly created layer is not added to other groups automatically.

Deleting a layer from a group does not make this layer deleted from the document. Deleting a layer from a document makes it automatically deleted from a group (groups).

By default, when a group is created, it is assigned a name made up from containing the word "Group" and the sequential number. This name can be edited.

Groups of layer properties are saved in the document.

Layers Filter

Filters allow creating dynamic groups of layers with properties that satisfy the filtering criteria. For example, the only layers of violet color, or the layers with names starting with "c", can be selected. The contents of the dynamic group changes along with the properties of the layers. For example, when creating a new layer, whose properties satisfy the filtering conditions, that layer will be automatically shown among the contents of the filter. Filters can be converted to layer groups (see *SectionConverting a filter into a group* on p. 1197).

Filters are named objects. By default, when creating a filter, at it is assigned a name consisting of the word "Filter" and a sequential number. This name can be edited.

Filters are saved in the document.

3.5.6.9.2. Managing Layer Groups

Creating Groups



To create a group, select its parent in the Sheets, views and layers tree and click the **Create group** or **Create properties group** button. These buttons are enabled depending on the grouping setting of the document (see Section 9.2.6.3 on p. 1760).

The icon of the created group will appear under the in its parent object. The text cursor will appear in the group name field. You can change the default group name. After pressing *<Enter>* or clicking the mouse outside the group name, it will be fixed.

- ▼ If inheriting parent item structure contents is enabled (see Section 9.2.6.3 on p. 1760), the new group will contain the layers. Otherwise, an empty group is created.
- ▼ If specific layers were selected in the parent object, the new group will contain these layers. This does not depend on whether contents inheriting was enabled or not.

To put the desired objects into the group, select a view, filter or a previously created group in the Tree of sheets, views and layers.

The Sheets, views and layers list will show the layers which make up the selected object.



You can limit the list of layers of the selected object using the Search area (see Section 3.5.6.2.4 on p. 1188).

Drag and drop the required layers onto the icon of the created group using standard Windows Drag&Drop option. This does not delete layers from their former previous position.



If an attempt to move the object to an invalid location is made, a warning sign will appear next to the cursor.



Several selected objects can be moved together. To select the items located sequentially, select the first of them and press *<Shift>*; then, while holding this key, select the last item. All objects between the first and the last one will be selected. To select arbitrarily located items, press *<Ctrl>*. While holding the key, keep clicking the object names. These selection techniques described above can be combined.

If a layer properties group is created, set the properties which each layer will receive after grouping. For this purpose, use the buttons of the toolbar (see Table 3.5.8 on p. 1184) and of the List of Sheets, Views and Layers field.

To change the name of an existing group, use standard Windows methods:

- ▼ click the group name,
- ▼ Highlight the name and press the *<F2>* function key,
- ▼ Call the context menu command **Rename**.

A text cursor will appear in the name field, allowing to edit it.



To delete a group, select it in the Tree of sheets, views and layers and click the **Delete** button.

Creating Filters



To create a filter of the layers of a document or its view, select the parent object in the Sheets, views and layers tree and click the **Create filter** button. This button is enabled if the layer grouping mode is enabled in the current document (see Section 9.2.6.3 on p. 1760).

The choice of the parent object for a filter determines the list of the objects to be filtered. If a filter is created at the view level (the second level), then filtering will be applied to the layers of the entire document. If a filter is positioned inside a view (the third level), then it will be applied only to the layers of that view.

A filter cannot be created within a group.

After clicking the **Create Filter** button, the **Layer filter** dialog will be displayed (Fig.3.5.23). The dialog's interface items serve to set up the filter. The text cursor will appear in the **Filter Name** field. You can change the default filter name. By default, the new filter is assigned a name consisting of the word "Filter" and a sequential number.

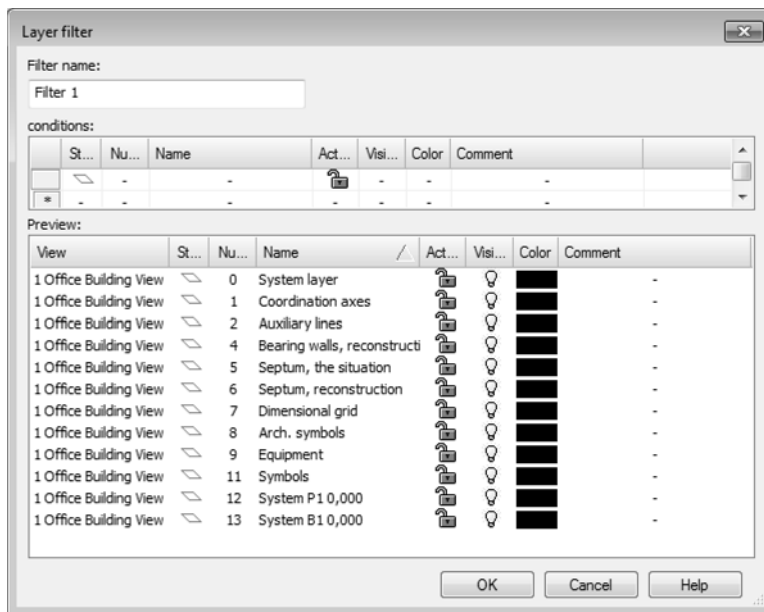


Fig. 3.5.23. Dialog for creating filters

Each line of the **Conditions** table is a condition of filtering layers. The **Status**, **Activity**, **Visibility** and **Color** fields contain drop-down lists. You should select a filtering condition from the list in those combo boxes pull-down menus. In the **Number**, **Name** and **Comment** fields you enter the conditions by typing from the keyboard. In this case, you can use masks. The "*" character used in masks replaces any arbitrary number of any characters. "?" character replaces any single character. To finish the input in a field, press <Enter> or click with the mouse outside the field.

When setting up conditions, you can use the context menu commands. The **Clear Cell** command serves to delete a value in the current field of the condition. The **Delete Row** command serves to delete a whole condition. The **Copy Line** command serves to create a copy of the current row.



There can be multiple filter conditions. After a filtering criterion is set, a new line is opened for creating the next condition. If a filter contains several criteria, they are united. For example, if the **Status** field of one condition contains the ‘non-empty layer’ value, while in another condition that is ‘empty’, then both empty layers and those containing objects will be shown in the filter layers and layers containing with objects.

The **Preview** table contains the list of layers, whose properties satisfy the filtering conditions. Before at least one condition is created, the list displays all layers of the filter’s parent filter object. The list changes dynamically according to the filter criteria change.

To finish up building a filter, click the **OK** button. The dialog will be closed. The filter mark will appear in the Tree of sheets, views and layers. To close the dialog and reject changes, click the **Cancel** button.

The filter name can be changed in the same way as the group name (see p. 1290).



To delete a filter, select it in the Tree of sheets, views and layers and click the **Delete** button.

Converting a filter into a group

If the mode of grouping layers is active in the document, then arbitrarily selected filters can be converted into groups of layers.

To convert a filter into a group, activate that filter and call its context menu command **Transform filter to group**. The object icon and name will change: the name will become the word “Group”, followed by a sequential number in the list of groups.

When changing the mode of grouping layers to the mode of grouping layer properties, all filters are automatically transformed into the groups of layer properties. The names of the resulting objects will not change in this case, i.e., that is, they will consist of the word “Filter” with the old sequential number.



Inverse conversion, i. e., converting a group into a filter, is impossible.

3.5.6.9.3. Using Groups for Setting the Layer Properties

Layer group

Using a group of layers serves to simultaneously set activity, visibility, ability to print or color of layers, which belonging to this group, in a similar manner. When a group is selected in the Sheets, views and layers tree, its contents are displayed in the Sheets, views and layers list. The fields in the **List** will reflect actual state of the layers in the group. Layer properties can be different from a layer to a layer. Using the buttons on the toolbar, set the state of visibility, activity, ability to print and color for the group.

Note that you cannot change activity and visibility of the current layer.

The group of layers may contain no current layers. The maximum number of current layers equals the number of views, to which the layers, contained in a group, belong. For example, if a group contains layers from View1 and from View2, then it can have two current layers. Thus, a group of layers, subordinate to a view, and also any group of fragment layers, may contain only one current layer.

To make the introduced changes of group elements effective, click the button **Apply**. In this case, the **Document Manager** will stay on the screen. If you click the **OK** button, then the settings will be applied to the layers in the group, and the **Document Manager** will close. To close this dialog without applying the introduced settings, click the **Cancel** button.

Filter

Using filters is similar to using a group of layers. When a filter in the **Tree** is selected, the layers with properties matching filter criteria, will appear in the Sheets, views and layers list. Using the buttons on the toolbar, you can define the property of being active, visibility, ability to print and color for of all filtered elements simultaneously. Using interface items in the fields of the List of Sheets, Views and Layers fields, you can modify its properties individually.



If the properties of one or more filtered layers were changed so that the layers do not match the filtering criteria any more, these layers are removed from the filter list.

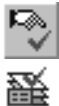
Layer Properties Group

Using a group of layer properties allows assigning an individual combination of properties to each layer belonging to the group. After a properties group is created, the values in the fields of the Sheets, view and layers list reflect actual layer states. Using the buttons on the toolbar or modifying layer property values directly in the fields of the List of sheets, views and layers, define the state of visibility, the property of being active, the ability to print and the color for each layer in the group. To assign the same set of properties to several layers, select them in the **List** and use the toolbar buttons in the toolbar.

Note that you cannot change properties of the current layer.

A group of layer properties allows either a single current layer or none at all.

Then, when the group is selected in the **Tree**, the properties assigned to this group's layers group will be shown in the Sheets, views and layers list. The actual condition of layers included in a group is not displayed.



To make the user-specified group element settings active, highlight the group of properties and layers and click the button **Make current**. A "checkmark" will appear on the group icon. Then, press the **Apply** button. In this case, the **Document Manager** will stay on the screen. If you click the **OK** button, then the settings will be applied to the layers in the group, and the **Document Manager** will close. To close this dialog without applying the introduced settings, click the **Cancel** button.

3.5.7. Technical specifications in a drawing

3.5.7.1. Entering technical specifications in a drawing

To start entering the text of technical specifications, select **Insert – Technical specifications – Input** in the main menu. The window designed for working with technical specifications will appear on the screen. Working with the dialog is described in Section 3.5.7.1.1 on p. 1199.

Perform the required actions and leave the mode. By default, technical specifications are displayed above the drawing's title block.

For more information about editing or deletion of technical specifications, see Section 3.5.7.4 on p. 1202.

3.5.7.1.1. Working with technical specifications

The window designed for working with technical specifications is used for entry, editing, and formatting of the respective text. This dialog appears on the screen after calling the **Insert — Technical Specifications — Enter...** command.

The window has the following header: *File name* —> *Technical Specifications*.

This window displays specifications if they are assigned to the document. You can also see the dotted frame along the borders of technical specifications on the sheet. By default, borders in the drawing coincide with the width of the title block. For details about changing borders, see Section 3.5.7.2 on p. 1200.

All text entry, editing and formatting options are available in this mode (see Section 4.1.2 on p. 1293).



Using the **Insert text template** on the **Insert** tab of the Property bar (see Section 4.1.2.11.6 on p. 1312) makes creation of technical specifications faster, since there is no need to re-enter paragraphs that are frequently used.



Using the **Save block** and **Load block** commands in the **Insert** tab of the Property bar, you can save and insert typical technical specifications into text – text blocks from text files (*.txt) or from KOMPAS-3D text documents (*.kdw). More information about using text blocks can be found in Section 4.1.2.10 on p. 1307.



Using the **Check Spelling** command in the **Format** tab of the Property bar, you can check spelling in different languages.



The entered technical specifications are transferred to the document after saving the changes made in the window.

The **File** menu contains special commands for saving and closing listed in Table 3.5.11.

Tab. 3.5.11. Special commands of the **File** menu in the window designed for working with technical specifications

Command	Description
Close — Document	Closes the document to which the specifications being edited belong. Specifications window closes, as well.
Close— Technical requirements	Closes the specifications editing window. The document's window, to which they belong, remains open.
Save — To document	Saves the specifications to the document they belong to. The document is not saved to disk.

Tab. 3.5.11. Special commands of the **File** menu in the window designed for working with technical specifications

Command	Description
Save — with Document to File	Saves the specifications to the document they belong to, and the document itself to disk. The specifications window remains active.
Save as — Text document	Saves the text of the specifications into a separate KOMPAS-3D text file (*.kdw). Upon calling the Save command, a dialog is displayed, in which you need to specify the folder and the file name for saving is displayed.
Save as — with Document to File	Saves the document to which the specifications belong, under a different name to another file or to another folder. The specifications window of the newly created document remains active after the command execution.

To close the window with BOM editing, you can also click the editing mode icon in the document window.

3.5.7.2. Placing the Specifications onto a Drawing

There are two ways to switch to manual mode of when placing specification on a drawing:

- ▼ select **Insert— Specifications— Location in the main menu**,
- ▼ select **Edit specifications** in the context menu,



These commands are enabled if the specifications text is already created (see Section 3.5.1.4 on p. 1150).

In manual placement mode, the specifications text is enclosed in a boundary with corner dots (see Figure 3.5.24).

To move the specifications to another position, point to the inside of the boundary (the cursor will become a four-sided arrow), click and hold the mouse while moving it. The specifications will move along with the cursor. Once the desired orientation is reached, release the mouse button.



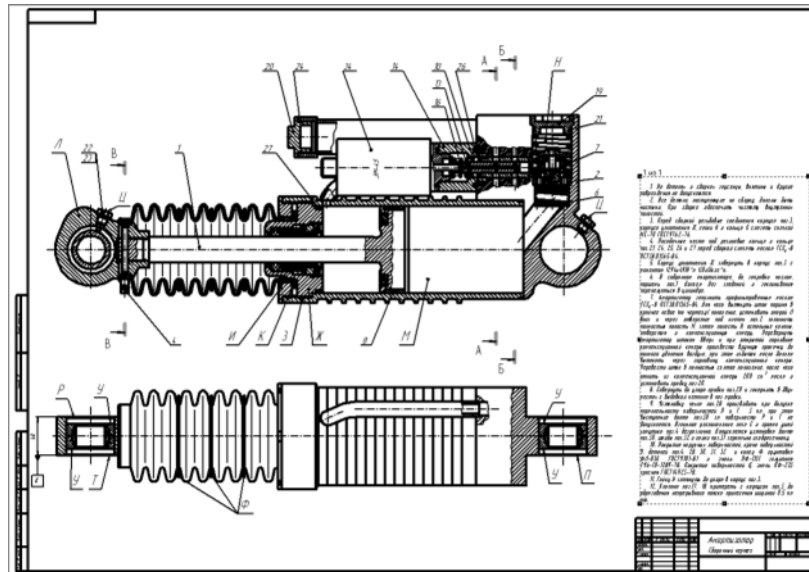


Fig. 3.5.24. Specifications placement mode

Drag the side and corner dots of the boundary to resize the text field of the specifications.

Exact position and size of the specifications page can also be specified. While in manual specifications placement mode, open the context menu and select **Locate...**

A dialog box will open for placing current specifications page. It allows to specify the distances from the left and bottom borders of the drawing (outer border) to the specifications boundary.

3.5.7.3. Breaking the Specifications into Pages

Specifications text is automatically placed above the title block of the first or the last (depending on the setup, see Section 9.2.6.18.6 on p. 1826) sheet. If the whole text does not fit into this space, it is automatically broken into pages which are shifted to the left.

If the text does not fit inside the drawing boundary, a larger page format should be chosen (see Section 3.5.2.1 on p. 1151). If the specifications are long enough to occupy ten or more pages (for large assemblies, for example), we recommend putting them on a separate sheet as a text document (for information on how to create text documents, see Section 4.1.5 on p. 1342).

If necessary, you may break the specification into pages manually and arrange them individually on the drawing.

Switch to manual specifications placement mode and decrease the vertical size of the specifications boundary (see Section 3.5.7.2). A new boundary (second page) will be created around the text that did not fit into original boundary (first page).

The second and all other pages of the specifications are placed on the vacant space of the sheet according to standard rules.

An example of specifications break-down into pages is shown in Figure 3.5.25.

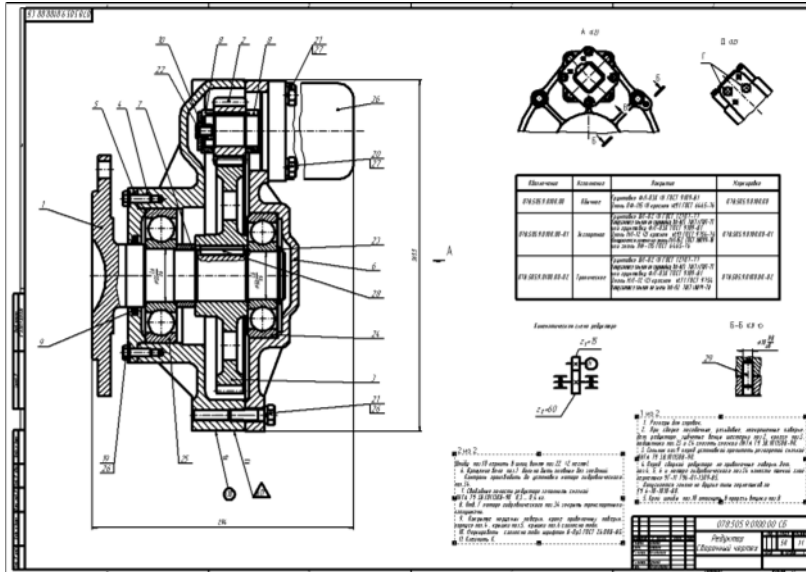


Fig. 3.5.25. Breaking the Specifications into Pages

The reverse operation is also possible. If you stretch the original boundary wide enough, the entire text will fit inside it, while additional boundaries will be removed.

To restore the default size and position of the specifications, select **Auto locate technical specifications** in the context menu. If a drawing occupies several sheets, the default position of the specifications depends on the settings of the sheets (see Section 9.2.6.18.6 on p. 1826).

3.5.7.4. Editing and Deleting Technical Specifications

There are three ways to switch to specifications editing mode:

- ▼ double-click the specifications,
- ▼ select **Edit specifications** in the context menu,
- ▼ select **Insert— Specifications— Input** in the main menu.

There are three ways of deleting the specifications from the drawing:

- ▼ Select **Edit— Delete— Specifications** in the main menu
- ▼ select **Edit specifications** in the context menu,
- ▼ delete specifications text while editing.

3.5.8. Breaking a Drawing into Zones

Auto save is set up in the dialog (Fig.) invoked with the **Service — Parameters... — Current Drawing — Document Parameters — Zoning** command (see Section 9.2.6.18.4 on p. 1824).

The zoning parameters are stored in the drawing itself and do not change when the drawing is moved to another workstation.

If you use the same zoning parameters for all drawings, it may be inefficient to set them individually for each document. The new drawings can be set up to be created with the required zoning parameters already in place.

For this purpose, the dialog is used which is called by the following command: **Service — Parameters... — New Documents — Graphic Document — Document Parameters — Zoning.**

This setting applies only to documents created after its implementation. These parameters will affect only the documents created after the settings were made. Drawings existing prior to this setting taking effect will not change.

3.6. Associated Views

3.6.1. General Information on Associated Views

Associated views are produced in a standard KOMPAS-3D drawing. A drawing that contains associated views is called associated drawing.

The following views are available:

- ▼ a standard view (front, back, top, bottom, right, left),
- ▼ arbitrary view (view of an arbitrary model with an arbitrary orientation),
- ▼ a projection view (a view in the direction specified with respect to another view),
- ▼ a view in arrow direction,
- ▼ cut/section (simple, with a jog, by a polyline),
- ▼ enlarged detail,
- ▼ detail view,
- ▼ detail section.

Commands for creation of associated views are grouped in the menus **Insert — Model View** and **Insert — Auxiliary View**. Buttons for calling these commands are located on the **Views** bar (Fig. 3.6.1).

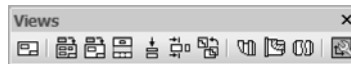


Fig. 3.6.1. **Views** bar

Standard and projection views are automatically constructed with the projection link.

All views maintain a relation with the model: model changes lead to image change in an associated view. While the associative view is linked to the model, it is impossible to edit geometric objects that make up the model's projection. An relation between a view and the model can be destroyed.

For any view the user can choose model objects not to be displayed on it. These may be bodies, surfaces, curves, points as well as layout elements created in the model (thread notations, dimensions, or other symbols). Besides that, displaying as a selected component view (i.e. all of its objects) can be disabled.

For a cut/section, user can appoint "uncut" components of an item (i. e., parts or subassemblies).

It is possible to synchronize data in the drawing's title block (designation, name, mass, and so on) with data in the model file, i.e. synchronize drawing's technical specifications with the model file's technical specifications.

A single drawing may contain several associated views, including views of different models.

If a model contains variants, you can choose the required variant when creating the associative view.

If there are any layers, layer groups or filters in the model, then you can copy them to the drawing when creating a new associated view (see 3.6.3.10 on p. 1235).

Views are easy to manage via the Drawing Tree (see Section 3.6.1.1).

Any associated view may include one or several image breaks (see Section 3.5.5.8 on p. 1177).

Commands to create a view in arrow direction, a cut/section or an enlarged detail will be called automatically after appropriate layout objects are created in a drawing: a view arrow, a cut/section line or an enlarged detail. These commands can also be called manually.

View captions along the arrow, as well as captions of cut/section and enlarged detail are automatically linked with their respective layout objects and thus will always match one another (see Section 3.5.4.6 on p. 1165).

The drawing containing associated views of a three-dimensional model is automatically assigned the attributes which contain information about the model mass (Fig. 3.6.2).

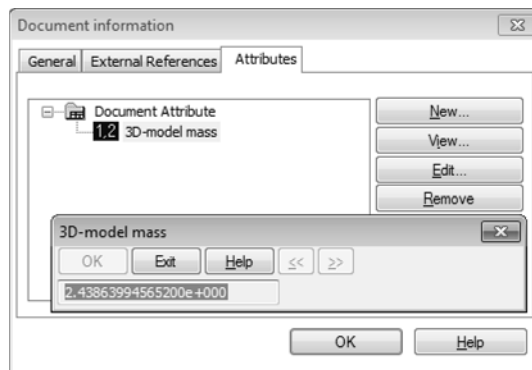


Fig. 3.6.2. Viewing info on a model in the drawing



If a drawing contains images of several models, then the information is entered in its attribute that refer to the model whose data appears in the title block of this drawing.



Associated views of an assembly or a multibody part can be displayed incorrectly if the components of an assembly or a part body intersect. The incorrectness consists in the absence of visible lines or the presence of invisible lines. To fix these errors, it is recommended to edit a model by eliminating the intersection of components or bodies. This can be done in one of the three following ways:

- ▼ to modify components or bodies by eliminating the excessive material,
- ▼ to modify the location of components or bodies so that they would not intersect,
- ▼ to combine intersecting components or bodies.

Errors in associated view may also be caused by the self-intersecting part bodies. To correct an image, you should edit the part to eliminate self-intersections of bodies.

3.6.1.1. Associated views in the Drawing Tree

Associated view automatically appears in the Drawing Tree right after it was created in the drawing. The name is also assigned automatically to the views. It contains the view name and its scale. For example, “System View (1:1)”, “Projection View 4(2:1)”, “Cut B-B (1:1)”, etc.

More about working with the Drawing Tree in Section 3.5.4.5 on p. 1163.

Subordinate objects of an associative view represent a model the image of which is contained in the view and local cuts, if they are present in the view (Fig. 3.6.3, a).

Objects that make up the model are grouped into sections: *Bodies*, *Surfaces*, *Curves and points*, *Components* (Fig. 3.6.3).



Model objects that do not take part in forming its image in the drawing, such as sketches, auxiliary axes and planes, LCS, and so on, are not displayed in the Drawing Tree.

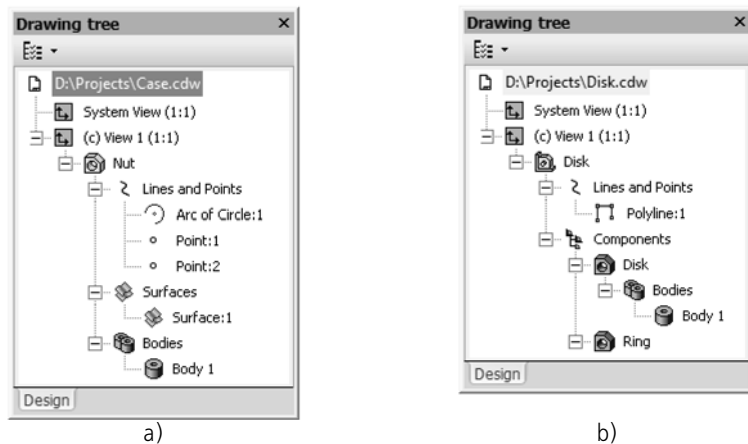


Fig. 3.6.3. Drawing trees with associative views:
a) parts, b) assemblies

Context menu of Drawing Tree objects (Fig. 3.6.4) makes it possible to see views' relations, as well as assign “uncuttable” components and switch to editing the 3D models shown in associative views.

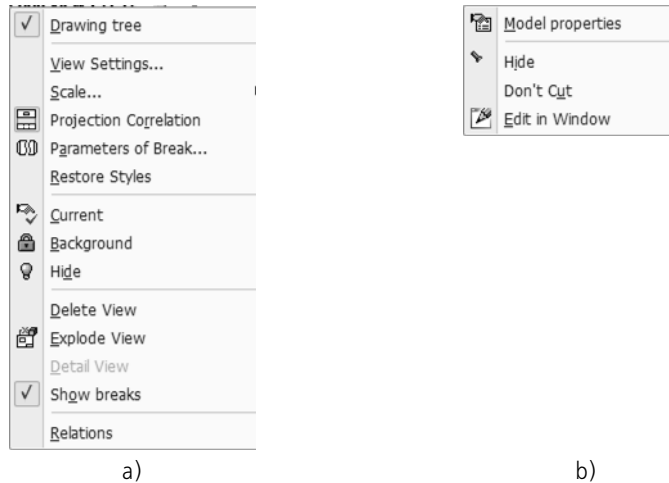


Fig. 3.6.4. Context menus of Design Tree elements:
a) view context menu, b) component context menu

Note the following specifics of a model object displaying in a Drawing Tree.

- ▼ In the *Surfaces* section, the surfaces that are **actually present in the model** will be displayed. The surface type is not defined. As a result, the contents of the *Surfaces* section in the Drawing Tree may be different from the contents of the same section in the Model Tree. For example, *Rotation surface:1*, *Linear surface:1* are present in a model together with the *Knitting* operation that combines them into a single surface. In the model's Drawing Tree, only one surface with the *Surface:1* name will be present. If a body was created in the process of knitting the surfaces, the Drawing Tree will not contain any *Surfaces* sections (despite the fact that it is present in the Model Tree).
- ▼ Some objects — bodies, surfaces, curves or points — may be hidden during the model editing. By default, such objects are not displayed in associative views. They are visible in the Model Tree, but their icons do not differ from visible objects' icons (unlike the Model Tree, where hidden objects' icons are gray). If necessary, you can enable displaying hidden objects in views (see Section 3.6.1.3.3 on p. 1213).

3.6.1.2. Standard Method of Creating a Model's Associated Drawing

1. Additional steps (if necessary).
 - 1.1. Create or open the model for your associated drawing.
 - 1.2. If you do not want that certain objects (components, bodies, surfaces, curves, points) appear on your drawing, disable their display in the model by choosing the **Hide** command.
 - 1.3. Specify any dimensions or symbols you want to transfer from the model to the drawing.
 - 1.4. If there are any layers in the model, specify the ones you want to copy to the drawing (for more information on model layers, see 2.15.13.2 on p. 846).

- 1.5. Choose the most suitable model orientation for the main view. If the desired orientation cannot be set using the standard orientations, then add a user orientation (see Section 2.1.3.2 on p. 121) that matches the desired one.
2. Create a new drawing in KOMPAS-3D.

For a simple model, you can use the standard construct view command to create its drawing (see Section 3.6.2.1 on p. 1216). This command allows immediately to immediately construct the entire set of required projections, including the isometric projection.

For assembly drawings and complex part drawings, we recommend using the command for arbitrary view construction (see Section 3.6.2.2 on p. 1218).



To automatically construct a drawing and an arbitrary view of the current model in it, select the command **Create new drawing from model** from **Operations** menu when working with the model.



If a model has unsaved changes, a request will pop up to save the model when you create its associated view. Click the **Yes** button to immediately save the model file or the **No** button to cancel saving. Anyway, the associated view will have the model image created in its current view.

3. Now proceed to the construction of projection views, cuts, sections, enlarged details, detail views and detail sections based on the standard or arbitrary views you have just created on the drawing (see 3.6.2.3 on p. 1218 — 3.6.2.8 on p. 1222)).
4. If required, edit the picture image in associated views:
 - ▼ specify the display, in views of objects including bodies, surfaces, curves, points and layout elements, notations of threads, dimensions, and other model symbols (see Section 3.6.1.3.3 on p. 1213).
 - ▼ specify “uncuttable” components (see Section 3.6.3.1 on p. 1224),
 - ▼ hide components that should not be visible on the drawing (see Section 3.6.3.2 on p. 1225).
5. Add necessary drawing layout objects: dimensions, technological symbols, texts, other elements (centerlines, center marks, etc.) (see Section 3.3 on p. 975). Before creating any of these, it is recommended to enable associativity and constraints for all added objects (see Section 7.2.1.7 on p. 1604). This will make the manually added dimensions, technological symbols, centerlines and other objects link to the picture of the model and track its changes. For instance, if a hole diameter in a part on the associated view is modified, then the diameter of the circle depicting this hole will also change, as well as the dimension value associatively linked to it. Additionally, in the constraints mode, links and constraints are automatically applied to objects such as cut/section lines, view arrows, enlarged detail designations. Therefore, for instance, a horizontal cut line aligned by center of the circle depicting a hole, will remain horizontal and follow the circle when this hole is moved in the model.

Also, to facilitate the subsequent work with the drawing, it is recommended to create new layers (for information about layers in graphical documents, see 3.5.6 on p. 1181) for various types of objects.

6. Compose the views on a drawing sheet (see Section 3.5.5.7 on p. 1174). If required, disable projection links between views (e. g., this may be required to place a view in arrow direction in an arbitrary position on the sheet).

3.6.1.3. Associated Views Setup

Associated views settings are divided into four groups:

- ▼ **Options** (see Section 3.6.1.3.1),
- ▼ **Lines** (see Section 3.6.1.3.2),
- ▼ **Objects, dimensions and process symbols** (see Section 3.6.1.3.3),
- ▼ **caption** (see Section 3.6.1.3.4).


Settings are available for all associated views except the **Detail View** and the **Detail Section**. Objects and layout elements cannot be customized in the **Enlarged Detail** view. The settings of these views are the same as the respective settings of their base views.

Lines, objects, dimensions and process symbols can be set up by default (see Section 9.2.6.18.1 on p. 1820).








3.6.1.3.1. Options

After selecting the command for creation of an associated view, the **Parameters** page of the Properties bar will display elements for view settings. Common elements for simple and associated views are shown in Table 3.5.3 on p. 1163. Elements of the **Options** tab that are specific for associated views are shown in Table 3.6.1.

Tab. 3.6.1. Elements for changing settings of associated views

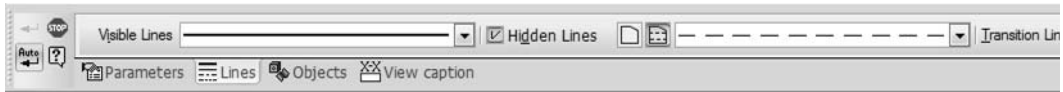
Element	Description
 Source File	<p>This field contains the full name of the model source file selected to appear in the drawing. To select other source file, enter its name into this field or use the button Select another model to the right from the field. After clicking it, the File Selection dialog will appear. Specify the required model in the dialog.</p> <p>If the model contains variants and additional numbers, the dialog will display lists for selection of the required variant and additional number. After choosing a source file, its full name will be displayed in the Source File field.</p> <p>Only for Standard and Arbitrary views.</p>
Version	<p>The list makes it possible to choose the model's variant for which the associative view is created. The view's geometry will correspond to the selected variant.</p> <p>The list is present in the Property bar, if the model, chosen for creation of the view, contains variants. Designations of variants, both dependent and independent, are located on the same level of the list's hierarchy.</p> <p>Only for Standard and Arbitrary views.</p>

Tab. 3.6.1. Elements for changing settings of associated views




Element	Description
Orientation model	This list allows to select a model orientation on the main view (when constructing standard views) or arbitrary view. The list contains all orientations from the model source file (on user orientations see Section 2.1.3.2 on p. 121). Only for Standard and Arbitrary views.
 Layout	The switch that allows to set up a standard view pattern for the model. Only for Standard views.
Select	The button for automatic selection of a standard scale for the drawing to fully contain all the views. Only for Standard views.
 Projection link	The switch controls the projective relation between associated views. Only for Arrow direction , Projection and Cut/section views.
 Explode	The switch to manage an assembly explosion. Available when the source file is an assembly with specified component explosion settings (see Section 2.15.7 on p. 836). Only for Arrow Direction , Arbitrary , Projection and Cut/Section views.
 Flat pattern	This switch serves to control display parameters of the flat pattern of a sheet body. It is accessible if the source file is a part containing a sheet body and flat pattern parameters (see Chapter 2.4.7.2 on p. 304). On displaying of a flat pattern in drawings, see also Section 2.4.7.4 on p. 309. Only for By Arrow , Arbitrary and Projection views.
 Layers	A switch to control the copying of the layers, layer groups or filters from the model into a drawing. Only layers with the Project into a drawing attribute can be copied; see Section 3.6.3.10 on p. 1235 for details. Only for Standard views and Arbitrary , Projection , In arrow direction , Cut/section views.
 Cut/Section	This is a group of switches that serves to select the fillet image type. Only for Cut/section and Detail section views.
	

3.6.1.3.2. Lines

After selecting the command for creation of an associated view, the **Lines** page of the Properties bar will contain control elements for setting the view display (Fig. 3.6.5). These elements are presented in Table 3.6.2.

Fig. 3.6.5. **Lines** tab

Tab. 3.6.2. Elements for changing the display of associated views

Element	Description
Visible Lines	The list to select a style* to display visible lines of the model projection.
Hidden Line	<p>This option controls the transfer of hidden contour lines to the view. If this option is disabled, then the hidden line display switch is inaccessible. Upon enabling the option, the system refers to the model and generates the hidden contour lines. The Hidden Lines display switch becomes accessible, and he can enable the hidden lines display.</p> <p>If the Hidden Lines option state was changed during the view parameters setup, then after finishing settings, this view is displayed crossed out, that is requiring a rebuild. As a result of rebuilding, the indivisible contour lines are added to the view (provided that the Hidden Lines option was enabled) or removed from it (if the option was disabled).</p> <p>It is recommended to disable the Hidden Lines option to the view when building associative views of assemblies with more than a 1000 components.</p>
 The Switch for Displaying Hidden lines	<p>This switch controls the display of hidden lines of the model projection. When the display is switched on, the list of line styles becomes available. By default, the dashed line is used.</p> <p>The Hidden Lines display switch is accessible if the Hidden Lines option is enabled.</p>
 The switch to control the display of tangent edges.	<p>The switch controls the line display of the tangent edges. When the display is switched on, the list of line styles becomes available. By default, the solid thin line is used.</p>
 The switch to control the display of bend lines.	<p>The switch controls the line display of the bend lines. When the display is switched on, the list of line styles becomes available. By default, the dash 2 dots are used.</p> <p>For the By arrow, Projection or Arbitrary views, the Bend lines switch is accessible if the flat pattern drawing option is enabled when setting the parameters for this view (see Table 3.6.1).</p> <p>For Cut/Section, Detail Section and Enlarged View the Bend lines switch is accessible if the flat pattern drawing is enabled among the parameter settings for the base view.</p>

* For details on line styles see Section 3.2.1.1 on p. 893.



Hidden lines are stored in a view regardless of whether their display is on or off. If those lines do not need to be displayed, disable the **Hidden Lines** option— this will reduce the size of the drawing file.

3.6.1.3.3. Objects, Dimensions and Process Symbols

After you call the create an associated view command in the **Objects** tab of the Properties bar, controls will appear that allow to specify what model objects and layout elements will be displayed in this view (Fig. 3.6.6).

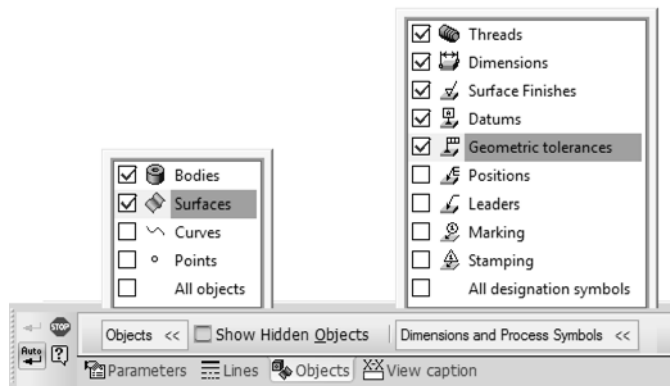


Fig. 3.6.6. **Items** Tab

- ▼ Enable the options in the **Objects** list that correspond to the object types to be transferred to the view.

Enable the **Show Hidden Objects** option to display the images of hidden objects along with visible ones. With the option disabled, the objects hidden in the model will not be shown in the associated view.

These options are applicable for all objects in a model, including the objects stored in the components.



Connection and control points are not transferred to a drawing.

- ▼ In the **Layout Elements** list, enable the options that correspond to the types of layout elements to be transferred to the view.

A cosmetic thread will be created in an associated view, if a projection plane of this view is strictly parallel or strictly perpendicular to the axis of the surface on which the thread is created.

A thread located on the invisible portion of the model is displayed in the associated view if the mode of showing hidden contour lines is enabled in it (see Table 3.6.2). The style set for those lines is also used to draw cosmetic threads.

The conditions for transferring dimensions and symbols to associated views are described in Section 3.6.3.5.1 on p. 1226.



Layout objects and elements that are included in hidden components (see Section 3.6.3.2 on p. 1225) are not displayed irrespective of the settings made in the **Objects** tab.



The model objects with the **Project into drawing** option set to **Forbidden** will not be added to the drawing even if they are enabled on the **Objects** tab.



The display settings for objects, dimensions and process symbols are unavailable for the **Enlarged Detail** view. This view displays the same objects as the objects presented in its base view, with the dimensions and notations that satisfy the transfer conditions.

3.6.1.3.4. Caption

After you call the create associated view command, the **View Caption** tab of the Property Bar will display the controls that allow to customize a view caption.

View caption is a text object included into the view. The caption contains both the links to the layout object associated with the view and links to the parameters of the view itself. For more information on view captions, see Section 3.5.4.6 on p. 1165.

The **View Caption** tab of the Property Bar is shown in Fig. 3.6.7, and the description of its controls is shown in Table 3.6.3.

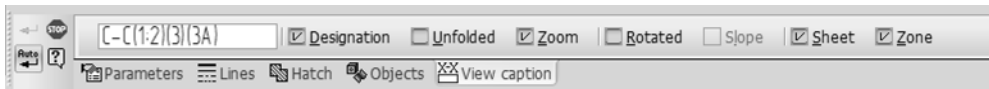


Fig. 3.6.7. **View Caption** tab

Tab. 3.6.3. View caption controls

Element	Description
View Caption Preview Window	The reference field containing the current view caption.
Designation	Disabling this option excludes a layout object symbol (for example, <i>B-B</i>) from the view caption. You must enable the Symbol option to include this text afterwards.

Tab. 3.6.3. View caption controls

Element	Description
Reoriented	<p>When the option is disabled, the “Reoriented” icon will not be included in the view caption. When the option is enabled, the icon will be included:</p> <ul style="list-style-type: none"> ▼ in the caption of an arbitrary view, projection view or view in arrow direction, if it displays a flat pattern of a sheet body (i.e., the Flat pattern switch is active in the Parameters tab of the Property Bar). ▼ in the caption of the view containing a cut/section or an enlarged detail if its base view displays a flat pattern of a sheet body.
Scale	This option allowing to include the current scale in the view caption.
Rotated	<p>When the option is disabled, the “Rotated” icon won't be included in the view caption.</p> <p>When it is enabled, the icon will be included in the view caption if the rotation angle set in the Parameters tab of the Property Bar is non-zero[*].</p>
Angle	The option allowing to include the rotation angle value in the view caption. The Angle option is available if the Rotated option is enabled.
Sheet	<p>The option allowing to include the link to the drawing sheet number, where the layout object associated with the view is located, in the view caption.</p> <p>The Sheet option is available if the Designation option is enabled.</p>
Area	<p>The option allowing to include a link to the drawing zone designation symbol, where the layout object associated with the view is located, in the view caption. If the option is enabled, the zone designation symbol will be added to the caption provided that zoning is enabled in the drawing. If the option is disabled, then the zone designation symbol is not added to the view caption.</p> <p>The Zone option is available if the Designation option is enabled.</p>

* If the view rotation is caused by the existing projective relation with its base view (the **Projection Correlation** switch is active in the **Parameters** tab), then the 'Rotated' mark is not included in the view caption.

3.6.2. Construction of views

Standard and **Arbitrary** views may be constructed regardless of existence of any other associated views in the drawing.

All other types of views must be based on the already existing views.

Views used to create other views are called **reference** views. For instance, for a projection view, the reference view will be the one used for the direction reference when constructing this projection view. For a cut or a section, the reference view will be the one that contains the section line, etc.

By default, the view has the same drawing (see Section 3.6.1.3.2 on p. 1211) and display (see Section 3.6.1.3.3 on p. 1213) settings as the base view.

If a view has a break, the break with the same parameters will be created in the projection view, view in arrow direction or cut/section created on its basis. For this purpose, it is necessary that the projection direction is perpendicular to the direction of image moving in the base view (see Section 3.5.5.8.2 on p. 1179).



With respect to automatic breaks, you can perform operations intended for breaks created using the **Break in View** command: control the image in the document, edit, delete it, and so on. Handling breaks is described in detail in Section 3.5.5.8 on p. 1177.

3.6.2.1. Standard Views



To construct standard model views, call the **Standard Views** command.

Select model source file in the dialog box.

If the selected model contains variants, you can specify designation of the required variant in the dialog. The view's geometry will correspond to the selected variant of the model.

Once the source file is selected, the drawing's window displays the ghost of the views' outline rectangles.



By default, the system offers to create three views: the main view, the front view, and the left view. To change the standard view set of the selected model, select the **View Pattern** switch on the **Parameters** page of the Properties bar.

The view pattern selection dialog will be displayed (Fig.3.6.8). Relative location of views on a diagram depends on the projection method. The projection method is selected during setting parameters for a new view, see Section 9.1.12.4.1 on p. 2047.

Controls on the dialog box for selecting view patterns are presented in Table 3.6.4.

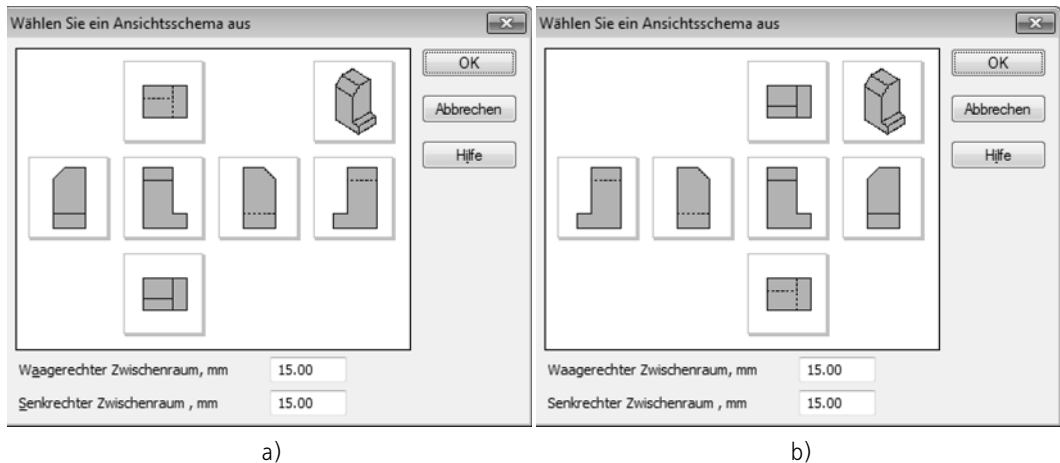


Fig. 3.6.8. View pattern selection dialog:
 a) projection method – **First angle projection**,
 b) projection method – **Third angle projection**

Tab. 3.6.4. Select View Scheme Dialog

Element	Description
View Pattern	Pattern of standard view arrangement on the drawing. Mouse-click a view to enable or disable its display. The main view cannot be disabled.
Gap Horizontal*	Horizontal distance between outline rectangles of views (in millimeters). This will could be the distance between e.g., the main view and the left view.
Gap Vertical*	Vertical distance between outline rectangles of views (in millimeters). This will could be the distance between e.g., the main view and the top view.

* The specified distances are measured “on paper”, without regard to the view scales.

Upon finishing the setup, click the **OK** button of the dialog.

Make settings (see Table 3.5.3 on p. 1163 and Tab. 3.6.1 on p. 1210) and views drawing (see Section 3.6.1.3.2 on p. 1211).

To fix the ghost, select a base point for the main view.



If a model has unsaved changes, a request will pop up to save the model when you create its associated view. Click the **Yes** button to immediately save the model file or the **No** button to cancel saving. In any case, the associated view will have the model image created in its current view.

3.6.2.2. Arbitrary View



To construct an arbitrary model view, call the **Arbitrary View** command.

Select model source file in the dialog box.

If the selected model contains variants, you can specify designation of the required variant in the dialog. The view's geometry will correspond to the selected variant of the model.

Once the source file is selected, the ghost of the outline rectangle will appear on the screen.

Set up view parameters (see Table 3.5.3 on p. 1163 and Table 3.6.1 on p. 1210), view drawing (see Section 3.6.1.3.2 on p. 1211) and the set of transferred objects (see Section 3.6.1.3.3 on p. 1213). Set up the view caption, if required (see Section 3.6.1.3.4 on p. 1214).

To fix the ghost, select a base point for the view.



If a model has unsaved changes, a request will pop up to save the model when you create its associated view. Click the **Yes** button to immediately save the model file or the **No** button to cancel saving. Anyway, the associated view will have the model image created in its current view.

3.6.2.3. Projection View



To construct a model view by selecting one of the orthogonal directions with reference to an existing view, select the **Projection View** command.

Select a base view to create the projection view. Then move the cursor into the direction that corresponds to the view you wish to create. Ghost view will be displayed.

Set up view parameters (see Table 3.5.3 on p. 1163 and Tab. 3.6.1 on p. 1210), view drawing (see Section 3.6.1.3.2 on p. 1211) and the set of transferred objects (see Section 3.6.1.3.3 on p. 1213). Set up the view caption if required (see Section 3.6.1.3.4 on p. 1214).

To fix the ghost, select a base point for the view.



The model's image in the final projection view depends on the selected projection method, see Section 9.1.12.4.1 on p. 2047. For example, if the **First angle** method is selected, move the cursor to the right relative to the reference point to form a view from the left. If the **Third angle** method is selected, move the cursor to the left.

If required, turn the view the other way around by changing the projection method (see Section 9.1.12.4.1 on p. 2047).

3.6.2.4. View in Arrow Direction

The arrow pointing the view direction in reference view is required to create a view in arrow direction. If the arrow is not present, create it (see Section 3.3.7.9 on p. 1037).



A necessary condition for a correct creation of a view in arrow direction is placing the arrow specifically in its base view. When creating a view arrow, ensure that an appropriate view is selected as the current view (see Section 3.5.5.2 on p. 1171 for view status control).

The **View in Arrow Direction** command is started automatically after view arrow has been created in the associative view.



If you have cancelled the execution of the automatically launched command, create the view in arrow direction by manually selecting the **View in Arrow Direction** command and then selecting a view arrow.

Ghost of the dimensional rectangle of a view will be displayed on the screen.

Set up view parameters (see Table 3.5.3 on p. 1163 and Table 3.6.1 on p. 1210), its drawing (see Section 3.6.1.3.2 on p. 1211), set of transferred objects (see Section 3.6.1.3.3 on p. 1213) and view text (see Section 3.6.1.3.4 on p. 1214).

To fix the ghost, select a base point for the view.

3.6.2.5. Detail View

A detail view depicts a separate and limited part of the model surface. In KOMPAS-3D, a detail view is constructed by trimming the image of an existing reference view. Any closed contour (circle, ellipse, closed Bezier curve, etc.) may be used as a trim limit.

If the reference view does not contain a closed contour, create it (see Sections 3.2.5, 3.2.6, 3.2.8, 3.2.9).



A necessary condition for a correct creation of a detail view is placing its bounding contour specifically in the base view. Therefore, when creating a contour, ensure that an appropriate view is selected as the current view (see Section 3.5.5.2 on p. 1171 for view status control).



To construct a detail view, select the **Detail View** command.

Select the contour that limits the detail view. Contents of the reference view outside the contour limits will disappear from the screen.

The full and the trimmed display of the view are switched by the **Detail View** command from the context menu of the view in the Drawing Tree.

3.6.2.6. Enlarged Detail

To create an enlarged detail view, it should be presented in the reference view of the enlarged detail. If the designation is not present, create it (see Section 3.3.7.11 on p. 1042).



A necessary condition for a correct creation of an enlarged detail element is placing its symbol specifically in its base view. Therefore, when creating a symbol, ensure that an appropriate view is selected as the current view (see Section 3.5.5.2 on p. 1171 for view status control).

The **Enlarged Detail** command is started after the enlarged detail's designation has been created in the associative view.



If you cancel the automatically initiated command, create the enlarged detail view by manually calling the **Enlarged Detail** command and specifying the enlarged detail designation.

A ghost of the outline rectangle of view that contains the enlarged detail will appear on the screen.

Set up view parameters (see Table 3.5.3 on p. 1163 and Table 3.6.1 on p. 1210), its drawing (see Section 3.6.1.3.2 on p. 1211) and text (see Section 3.6.1.3.4 on p. 1214).

To fix the ghost, select a base point for the view.

3.6.2.7. Cut/Section

To construct this view, its reference view must contain a section line designation. If the designation is not present, create it (see Section 3.3.7.10 on p. 1040).



A necessary condition for a correct creation of a cut/section is placing Cut/section Line Designation symbol specifically in the base view. Therefore, when creating a symbol, ensure that an appropriate view is selected as the current view (see Section 3.5.5.2 on p. 1171 for view status control).

The **Cut/section** command is started after the cut/section line has been created in the reference view.



If you cancel the automatically initiated command, create a cut or section by manually calling the **Cut/Section** command and specifying the cut/section line.

A ghost of the outline rectangle of view that contains the cut/section will appear on the screen. Set up view parameters (see Table 3.5.3 on p. 1163 and Table 3.6.1 on p. 1210), its drawing (see Section 3.6.1.3.2 on p. 1211), set of transferred objects (see Section 3.6.1.3.3 on p. 1213) and view text (see Section 3.6.1.3.4 on p. 1214).

To set up the hatches, select the **Hatch** page of the Property bar.

If the **Parameters from model** option is enabled, hatch parameters are automatically obtained from bodies and components of the model with respect to which the view is created (default variant). In this case, each component or body may have its own hatch parameters. Refer to Section 2.14.6.6 on p. 809 for more information about models' hatch settings.

If the option is disabled, control elements become available for manual adjustment of hatch parameters. Once all parameters are entered, the hatch of all components and bodies becomes uniform.

For more information about using the model-defined hatch, refer to Section 3.6.2.7.1 on p. 1220.



The **Hatch assembly** group of switches makes it possible to specify the method of hatching adjacent parts in the assembly: towards the same side or with a degree rotation 90°.



For other controls of the **Hatch** tab, see Table 3.2.7 on p. 959.

To fix the ghost, select a base point for the view.

3.6.2.7.1. Peculiarities of using the model-defined hatch

When creating associative views containing a hatch defined for a model, the following should be taken into account.

- ▼ Model hatch parameters are set up using the **Cut/Section** or **Detail Section** commands of the **Hatch** tab in the Property bar.

- ▼ Hatch parameters of components and bodies are transferred from model to drawing once – when creating the first associative view (irrespective of the **Hatch** tab settings). Parameters are saved in the drawing and can be used later for creation of new associative views with respect to this model. It means that hatch parameters for one component or body will be the same in all associative views.
- ▼ Later, when creating further associative views using one model, hatching parameters are transferred to the drawing only for new components and bodies which have no respective parameters in the drawing yet.
- ▼ Changing hatching parameters in the model does not result in their automatic update in views when the drawing is rebuilt.

In order to synchronize parameters of the model and drawing, use the **Restore Styles** command (see Section 3.6.3.4 on p. 1226).

- ▼ Hatching parameters can be changed manually in one or several associative views. For this purpose, highlight the hatch(es) and call the **Change style...** command from the context menu or from the **Service** menu.

After calling the command, hatch style adjustment dialog appears on the screen (Fig. 3.6.9).

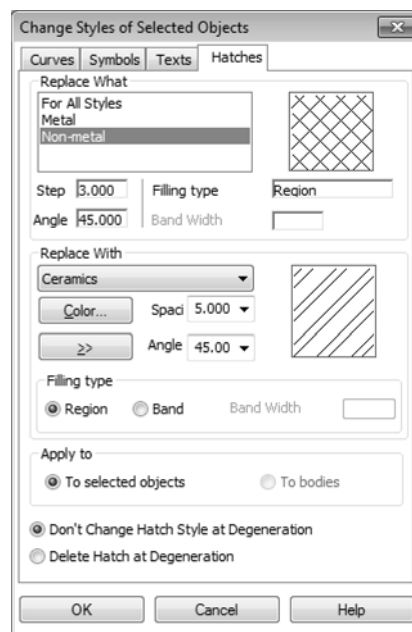


Fig. 3.6.9. Hatch style change dialog

In the **Hatch** tab, choose hatches that should be replaced in the **What to replace** list. In the **Replace by** list, choose a style for replacement that is available in the document. In order to choose a style for replacement from the library, click (**>>>**).

Specify color, step, and hatch angle, as well as other parameters, if needed.

In the **Apply to** group choose the parameter of application for the new style. By default, the **To selected objects** option is enabled. It means that the style will be changed for selected hatches only.

The **To bodies** option is available, if at least one hatch with parameters obtained from model is selected. Of the **To bodies** variant is chosen, styles of selected hatches will be replaced. At the same time, styles of non-selected hatches in all associative views will be changed for bodies with styles falling within the selection group.

3.6.2.8. Detail Section

To construct this view, its reference view must contain a closed contour (circle, ellipse, closed Bezier curve, etc.).

If the reference view does not contain a closed contour, create it (see Sections 3.2.5, 3.2.6, 3.2.8, 3.2.9).



A necessary condition for a correct creation of a detail section is placing its contour specifically in its base view. Therefore, when creating a symbol, ensure that an appropriate view is selected as the current view (see Section 3.5.5.2 on p. 1171 for view status control).



The contour shall not have self-intersections.



Creating of detail sections on enlarged details is unavailable.



To construct a detail section, call the **Detail Section** command.

In the base view, select a closed contour limiting the Detail Section (Fig. 3.6.10, a).

If necessary, enter the number and name of the detail section in the same-name fields on the Property bar. The **Cutting Plane** button is available when auto object creation is off (see Section 1.4.2.10), and also when editing an earlier created Detail Section. It serves to select the cutting plane position again.



Cut/Section group of switches serve to select the type of a view: a cut or a section.



The **Cut/Section** group will be available, if the base view of the constructed detail/section is not a cut (section) itself. Otherwise, the image type is the same as the type of the image on the base view, and the change is impossible.

To set up the hatches, select the **Hatch** page of the Property bar.

If the **Parameters from model** option is enabled, hatch parameters are automatically obtained from bodies and components of the model with respect to which the view is created (default variant). In this case, each component or body may have its one hatch parameters. Refer to Section 2.14.6.6 on p. 809 for more information about models' hatch settings.

If the option is disabled, control elements become available for manual adjustment of hatch parameters. Once all parameters are entered, the hatch of all components and bodies becomes uniform.

For more information about using the model-defined hatch, refer to Section 3.6.2.7.1 on p. 1220.



The **Hatch assembly** group of switches allows to select a hatching method for adjacent parts. For other controls of the **Hatch** tab, see Table. 3.2.7 on p. 959.

After specifying a limiting contour for the detail section, the system will display a ghost of a straight line which is a trace of the cutting plane. Note that this ghost is only visible when the cursor passes over a view with projection plane perpendicular to the projection plane of the constructed detail section.

Specify the cutting plane position (Fig. 3.6.10, b).

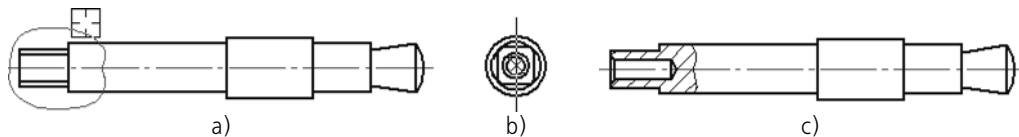


Fig. 3.6.10. Constructing a detail section: a) specifying a contour that limits the section, b) specifying the cutting plane position, c) result of command execution

The detail section will be constructed (Fig.3.6.10, c). Its icon will be displayed in the Drawing Tree “branch” that corresponds to its reference view (Fig.3.6.11).

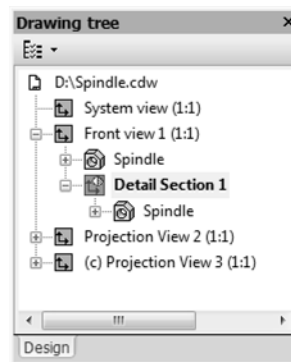


Fig. 3.6.11. Detail section icon

The context menu of a detail section in the Drawing Tree contains commands to control this detail section. They are represented in Table 3.6.5.

Tab. 3.6.5. Commands to control a detail section

Command	Description
Detail Section	It controls the Detail Section display on its base view. When a detail section is hidden, its limiting contour will still be displayed on the screen. To hide this contour, you may move it to another layer and then hide that layer.
Parameters of a section...	Serves to set up Detail Section parameters. After calling the command, the Property Bar displays the same controls as when creating an object.
Delete Cut	Serves to delete a detail section. This command displays the dialog that allows to confirm or revoke the view deletion. When a detail section is deleted, its limiting contour will remain in the drawing. It should be deleted separately.

3.6.3. Methods of Working with Associated Views

When working with associated views, you can use the general methods (see Section 3.5.5 on p. 1170) and a number of special methods.

3.6.3.1. Purpose of “Inseparable” Components

Immediately after the creation of a cut/section of a model with components (including the detail cut/section), all parts in the cutting plane will be displayed as cut. However, some parts should not be displayed as cut though the cutting plane intersects them. These are shafts, fixtures, etc.

To specify a part or subassembly that should not be cut, complete the following steps.

1. Expand the Drawing Tree branch corresponding to the view (cut or section) which should show an uncut component. Open the list of components and find the required component.
2. From the context menu of this component, select the **Do not cut** command.
3. The view containing the image of this component will be red-ticked in the Drawing Tree. This means that this drawing should be rebuilt.



4. Press the <F5> key or the **Rebuild** button on the **View** toolbar.

The image in the drawing will be rebuilt taking into account the introduced changes.

To re-enable the cut component display, call the command **Cut** from the context menu of its respective item in the Drawing Tree, and then rebuild the drawing.

Displaying of an uncut component in one cut (section) has no effect on its displaying in other cuts (sections). In other words, the component shown in several cuts (sections), may be shown cut in some of them, while in others — intact.

3.6.3.2. Hiding Component in View

To explain the internal design of an item, some of the views of its drawing would not show certain parts of the item.

It is entirely unnecessary to edit the 3D model to produce the image of an item without certain parts.

After having created the required views in a drawing (by default, they contain images of all visible components), you can hide any “unnecessary” components on some of them.

To do this, perform the following actions.

1. Expand the drawing tree branch corresponding to the view in which you need to hide any component. Open the list of components and find the required component.
2. From the context menu of this component, select the **Hide** command.
3. The view containing the image of this component will be red-ticked in the Drawing Tree. This means that this drawing should be rebuilt.



4. Press the <F5> key or the **Rebuild** button on the **View** toolbar.

The image in the drawing will be redrawn taking into account the introduced changes.

Hidden component icons are displayed gray in the Drawing Tree.

Hiding a component from display on one of the views does not in no way affects the display of that component on other drawing views.

To re-enable the component display, call the command **Show** from the context menu of its respective item in the Drawing Tree, and then rebuild the drawing.



If a component's displaying is enabled, its objects and layout elements are displayed in a view in accordance with the setup made in the **Objects** tab of the Property bar (see Section 3.6.1.3.3 on p. 1213).

3.6.3.3. Disabling a Projection Link between Views

Drawing projection views that were created by the **Standard Views** command are projection-linked to their main view; views created by the **Projection View**, **View in Arrow Direction** and **Cut/Section** commands are projection-linked to reference views.

The existence of protective relations between views limits their relative relocation. For example, a View by Arrow can move around the drawing field only in the direction indicated by the view arrow.

If you need to arbitrarily position some view, call the **Projection Correlation** command from the context menu of the view in the Drawing Tree. This command is also available from the context menu of a view selected in the drawing window. By default, the projection correlation is enabled (the **Projection Correlation** button is enabled). Call the respective command in order to disable the projection correlation., After that you can move a view in the drawing window with the mouse.



Another way to control a projective relation is as follows. Call the **View Settings** command from the context menu of the projection view in the Drawing Tree. The Property Bar will display

controls for setting up view parameters. Activate the **Projection Correlation** switch. After that, you can enter the view anchor point coordinates in the fields on the Property Bar.

To enable a project of relation, from the context menu of that view in the Drawing Tree, call the **Projection Correlation** command. A project of relation will be enabled.

3.6.3.4. Controlling line, point and hatch styles

By default, objects have the following styles in the associative view:

- ▼ body and surface projection lines — the styles set when customizing the view (see Section 3.6.1.3.2 on p. 1211),
- ▼ projections of points and curves — the styles of related objects in the model (see Section 2.5.4.1 on p. 361),
- ▼ hatches on cuts/sections — the styles set in the model for components and bodies (see Section 2.14.6.6 on p. 809).

If necessary, you can assign any other styles to lines, points and hatches (see Section 3.2.1.1.2 on p. 894).

Afterwards, in order to restore default styles of lines, points and hatches, call the **Restore Styles** command from the context menu of the view selected in the Drawing Tree or in the drawing. The command can be applied to several selected views at the same time.

3.6.3.5. Working with Projection Designations

A 3D model of an item may contain dimensions and designations (for details on how to create these objects, see Sections 2.9.2 and 2.9.3). Dimensions and designations may be transferred to a drawing containing associated views of the model. Objects appearing on the drawing as a result of this transfer are called **projection designations**.

3.6.3.5.1. Conditions for Forming Projection Designations in Views

If the transfer of dimensions and designations is switched on when setting the view parameters (see Section 3.6.1.3.3 on p. 1213), then the corresponding projection designations will be formed in the view upon the following conditions.

- ▼ The projection plane of the view is parallel to the plane of dimension or designation.
- ▼ Objects (components, bodies, surfaces, lines) of dimension or designation are displayed in a view. For example, position reference numbers shall be displayed in a view, if it contains visible edges or parts of faces, from which position leaders extend.

To form projection designations on sections, cuts, detail views and enlarged details, additional conditions must be satisfied.

- ▼ Dimensions and designations are transferred to a section only when their plane is coincident with the section plane.
- ▼ Transfer of a dimension or a designation to a cut or an enlarged detail depends on position of points specified as snap points (see Table 3.6.6).
 - ▼ Dimensions and designations are transferred to a cut only if none of the snap points are cut off by the cutting plane.

- ▼ Dimensions and designations are transferred to an enlarged detail or a detail view only if the projections of all their snap points are located inside the contour limiting the view.

Tab. 3.6.6. Dimension and designation snap points

Object	Fixing Points
linear dimension, linear dimension from line to point, angular dimensions	starting points of extension lines
radial dimension, diameter dimension	center of measured circle or arc
leader, position reference number, markings, stampings, surface finish symbols on leader shelves, datums, geometric tolerance	ending points of branchings
surface finish symbol without shelf	symbol location point

Note that projection designations that correspond to the linear and angular dimensions are formed in an associated view even when projections of their snap points are covered by visible lines, like on Fig. 3.6.12.

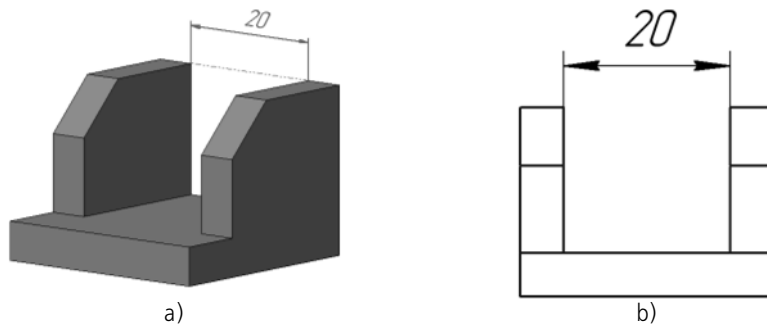


Fig. 3.6.12. Linear dimension in model (a) and corresponding dimension in associated view (b)



Projection designations of drawing views can be shown or hidden (see Section 3.6.3.5.2).

3.6.3.5.2. Controlling the Display of Projection Designations

Display of projection designations on a drawing can be enabled or disabled.

To hide visible projection designations, select them and then select the **Hide** command from the context menu. The selected projection symbols will be hidden from view.

To show hidden projection designations, do the following:



1. Select the **View — Show Hidden Designations**.

All hidden projection designations will be shown in light black on the screen. Shape of cursor will be changed.

2. Click on each of the designations you want to make visible.

When the **Show Hidden Designations** command is selected, you can also hide visible designations by clicking on them.



3. Finish the command by selecting the **Interrupt Command** button on the Special Control Panel.

Showing/hiding a projection designation on a view does not affect the display of the corresponding object in the model. And vice versa, showing/hiding a dimension or a designation on the model does not affect the display of the corresponding projection designation on an associated view of that model.

When creating a view on the drawing, some of projection designations on this view may be hidden automatically. This happens in the following cases.

- ▼ The corresponding dimensions or designations are hidden in the model but not excluded from calculation (objects excluded from calculation are not transferred to the drawing).
- ▼ Such projection designations are already present on other views of the drawing. For example, constructing an enlarged detail involves creation of projection designations that satisfy the forming conditions (see Section 3.6.3.5.1) but not displayed because they are already shown in the previously constructed reference view.

Besides, dimensions based on hidden contour lines are automatically hidden (even if the display of these lines in a view is enabled).

If required, you can enable the display of automatically hidden projection designations, as described above. This will not affect the display of projection designations (including duplicates) on other views.

3.6.3.5.3. Properties of Projection Designations

When working with projection designations, the following features should be considered.

- ▼ Projection designations have an associative link to their corresponding dimensions and designations in the model. Because of this link, the text and some other parameters of a projection designation (type of angular dimension, type of surface finish symbol, etc.) are transferred from the model to the drawing. A drawing is rebuilt when it is opened or by the command **View — Rebuild**.



Text of a projection geometric tolerance symbol is transferred from the model to the drawing when an associated view is created. It will not be refreshed when the drawing is rebuilt.

- ▼ The presence of an associated link between the model and a projection designation is characterized by the *Projective* constraint of this designation. If required, you can delete this constraint (see Section 7.2.3.1 on p. 1618). Then the projection designation will become an ordinary dimension or designation.
- ▼ Projection dimensions are linked not only with dimensions in the model, but also with geometric objects of associated type (i. e. with projections of the objects to which they are set). This

allows updating of the dimensions in accordance with changes in the model image when rebuilding the drawing.

- ▼ A letter of a projection designation of a datum is transferred from a model to a drawing only if you disable the auto-sorting of letter symbols in the drawing (see Section 3.3.12.20 on p. 1067). If the auto-sorting is enabled, then the projection designation of a datum will be assigned to the first unassigned letter in accordance with the sorting order. If the auto-sorting in the drawing is disabled, then the projection designation of datums will contain the same letters as at the moment it has been disabled.

3.6.3.5.4. Editing Projection Designations

Editing options for projection designations are limited by the presence of the link with the model.

Editing Parameters

To edit parameters of a projection designation, double click on it.

The Properties tab will display the same control elements for project parameters as displayed when creating ordinary objects of the respective type on the drawing.

Editing a projection designation has the following distinctive feature: parameters transferred to the drawing from the model cannot be edited. The other parameters are available and can be changed in any way you want. Changes are stored in the drawing. They are not updated when the drawing is rebuilt.

For example, when editing a linear dimension, all parameters will be available except the snap points coordinates and the orientation type; when editing a dimension text, the only available options are to underline the text, to put it into a frame or into parentheses, to hide the tolerance, and to select the location type for extreme values and deviations. You can also disable the auto determination of the dimension and enter any value you want.

Editing with Handles

When a projection designation is selected by mouse-clicking, its defining points will be displayed on the screen. By moving these points with the mouse, you can edit the configuration of this projection designation.

Using the defining points, you can, for example, change the position of the dimension line for a projection dimension or a shelf starting point location for a projection leader.



When editing the defining points of projection designations for datum, surface finish, geometric tolerance, marking, stamping, position and leader, you can also move the projections of their insertion points (see Table 3.6.6 on p. 1227). If the location of a designation insertion point projection has been edited manually in the drawing, then any subsequent changes of this location in the model will not be transferred to the drawing.

3.6.3.6. Conversion to model editing

If you have the main KOMPAS-3D module installed at your workplace - the 3D modeling system - and the electronic protection key has a license for its use, you can quickly open a document

containing the 3D model depicted on the drawing you currently work on. From an assembly drawing, you can convert directly to parts models.

To do this, expand the branch of the Drawing Tree corresponding to any view, which depicts the component (a part or a subassembly) to be edited, and from the context menu of that component call the command **Edit in window**.

KOMPAS-3D will open a window with the selected three-dimensional model. Make the desired changes, save and close the window.

When returning to the drawing window containing the image of the edited model, the system will prompt you to rebuild the drawing. The answer Yes to this query means rebuilding of the drawing image while accounting for the changes introduced in the model.

Rebuilding a model drawing consisting of a large number of parts and subassemblies can take a significant time. Therefore, if you need to edit several components of such a model, it is recommended to rebuild its drawing just once — after finalizing changes (rather than each time after editing an individual component).



Generally speaking, you can proceed with editing a component in the window by opening its file in the usual way: click the **Open document** button or call the command **File — Open** and select the desired file. When selecting the **Edit in Window** command, you do not have to search the disk folders for the proper file, as the system finds it automatically.

3.6.3.7. Warnings on Required Drawing Rebuilding and on Errors

Links between the model and its images allow to continuously control their matching relevance between them.

Every time when a drawing containing associated views is opened and activated, the system checks the relevance between the image and the model. If any discrepancies are found, the system will request the rebuilding of geometric objects on the drawing.

The positive answer to this query immediately starts rebuilding the drawing image. The negative answer postpones the rebuild. In this case, the view icons which images do not correspond to models are marked with a red “check” in the Design Tree with this meaning you need to rebuild the drawing. Such views are displayed crossed out in the drawing window.



You can rebuild the drawing at any time by using the <F5> key or the **Rebuild** button on the **View** bar.



If a model has unsaved changes, a request will pop up to save the model when you rebuild its associated view. Click the **Yes** button to immediately save the model file or the **No** button to cancel saving. In any case, the model image and title block contents in an associated view will correspond to its current view.

Rebuilding a drawing is also required after disabling/enabling the components display, assigning “inseparable” components, and in certain other cases.

If the model icon is checkmarked in the Drawing Tree, then a rebuilding is required for the model depicted in the view, and not for the drawing. To rebuild a model, open it (see Section 3.6.3.6 on p. 1229) and press the <F5> key or click the **Rebuild** button. Save the model and close its

window.

Sometimes, upon rebuilding a drawing, an error sign (an exclamation sign in a red circle) appears in place of a “check” thereby signifying an error in building a view. For instance, editing a cut/section line caused it to self-intersect. A cut matching such a cut/section line cannot be constructed, therefore, it's icon is marked with an error sign in the Design Tree.

To find out the error cause, highlight the error-containing object in the Drawing Tree, then select the **What's wrong?** command in the context menu.

A dialog in which the errors are listed, that occurred during this object rebuild will be displayed.

The help system contains recommendations on possible error-fixing techniques.

To get explanation for the specific error and general recommendations on fixing it, select the error description in the dialog and click the **Help** button.

If a view contains an image of an erroneous model, then a model icon, a view icon and a drawing sheet icon shall get an error sign in the Drawing Tree. Due to this, the presence of an error in the model is noticeable even if the Design Tree is fully collapsed.

To fix an error in a model, open and edit it following recommendations of the Help System.

3.6.3.8. Filling the Title Block of Associated Drawing

When creating a **Standard** or **Arbitrary** view, the following data are transferred from the model document to the drawing Title Block:

- ▼ marking,
- ▼ name,
- ▼ mass,
- ▼ material (only for part drawings).

These data are entered when working with the model document (**.m3d*, **.a3d*) while setting the properties of a part or an assembly.

The part mass is calculated automatically according to the material density and part shape. The assembly model mass is defined as the combined mass of its components.

If the drawing contains associated images of several models, then you can may select a model document to be used as a source for filling the Title Block.

To do this, double-click on the drawing Title Block, thus entering the edit mode.

Select the **Models** page of the Properties tab. This page displays the lists of model documents with associated views contained in the active drawing.

The name of the model whose data are transferred to the drawing Title Block is marked by a drawing sheet icon to the left of it. To specify another model to be linked with the Title Block, select it in the list and click the **Connect** button (or double-click the required line).

The text in Title Block fields will change. To confirm the model selection, choose the button **Create Object**. The system will return to the normal mode, and the Title Block will contain data on the model specified.

Filling the other fields of the Title Block is similar to filling Title Block fields of an ordinary drawing (see Section 3.5.3.1 on p. 1155).



Information is also transferred from the document model to *Developed by*, *Examined by*, *Approved by*, *In-process inspection*, *Quality inspection* columns, if values of the respective system properties are set in the model. For details, see Section *Model properties in the associated drawing* on p. 1388.

3.6.3.8.1. Synchronizing the Title Block and the Model

The link between a model and the Title Block of the drawing that contains its associated views allows to **synchronize** the Title Block and the model.

Synchronization between the Title Block and the model is a data transfer between the model properties and the drawing Title Block.

For example, after creating associated views of a part on a drawing, we need to change its material of production. To include the new material specification and the new mass to the drawing Title Block, we need to synchronize the Title Block and the model.

The data that may be transferred from a model to a drawing are the same as those when creating associated views:

- ▼ marking,
- ▼ name,
- ▼ mass,
- ▼ Material.

Data transfer from the model to the drawing is set up and controlled from the **Parameters** page of the Properties tab in the drawing title block editing mode. Control elements of this page are shown in the Table 3.6.7.

While working with a drawing, you can at any time synchronize the data in the drawing title block and in the model properties.

Thus, to get the data from the model (designation, name, material and mass), call the **Synchronize Data** command from the context menu of the title block in the filling mode.







If model associative view was copied by clipboard to new drawing, then the view should be rebuilt to transfer data from this model to drawing title block.

Only the product designation and name can be propagated in the opposite direction — from the drawing to the model. This transfer does not require a setup and is performed automatically when the drawing is saved. Standard document code and name (if any in the drawing title block) do not get into the model.



List of standard document codes and names is stored in the code and name file *graphic.kds*. By default it is stored in \Sys subfolder of the main system folder. File format— is a text one that is why if required *graphic.kds* file may be opened and edited in any text editor.

Tab. 3.6.7. Control elements for title block synchronization

Element	Description
 Synchronization of data	The switch to enable or disable the Title Block synchronization with the model. If active, data transfer is enabled both from model to drawing and from drawing to model. If not active, data are not transferred in any direction.
Synchronize	The button to obtain data from the model (designation, name, material, mass).
Mode synchronization	Group of switches for synchronization mode selection.
	If the Auto Synchronize switch is active, then the data will be transferred from the model to the drawing Title Block on opening and activation of associated drawing or on rebuilding the image.
	If the Synchronize By Query switch is active, then only the manual synchronization is possible*.
 Synchronize Mass	This switch controls the mass synchronization of the model. When active, the model mass will be recalculated and displayed in the <i>Mass</i> field on every synchronization. If the switch is inactive, the mass will not be calculated, and its value in the title block will not change.
Precision of representation	The field to input or select the number of decimal digits in the display of mass value.
Mass	The list allowing to select measurement units to display model mass value in the title block: kilograms, grams or tons .
Abbreviation	The field for the mass units designation. Symbols entered in this field are transferred to the <i>Mass</i> field.

* To do this, you should select the **Synchronize** button on the Properties tab or the **Synchronize Data** from the context menu of the Title Block. If a model has unsaved changes, a request will pop up to save the model. Click the **Yes** button to immediately save the model file or the **No** button to cancel saving.

You can set up the synchronization of a drawing's and a model's title blocks in the dialog which is called by the **Service — Parameters... — Current Drawing — Document Parameters — Main text** (see Section 9.2.6.18.2 on p. 1823).

Control elements from this dialog have the same functions as the elements from the **Parameters** page shown in Table 3.6.7. The only difference is the absence of the **Synchronize** button in the dialog as it is not used during setup.

3.6.3.9. Transfer of technical specifications from model to drawing Synchronization

Technical specifications can be transferred to the associative drawing from the model for which a drawing has been created. Links to model objects in the transferred specifications will represent links to the respective objects in the drawing. Afterwards, if technical specifications in the model are edited, changes may be transferred to the drawing by means of their synchronization. Transferring of technical specifications from a drawing to a model is not supported.

Refer to Section 2.15.11.1 on p. 843 for more information about entry of technical specifications in the model.

Specifications are transferred to the drawing automatically during creation of the associative view if all of the following conditions are met

1. The **Synchronize technical specifications** option in the dialog of technical specifications parameters is enabled (see Section 9.2.6.18.6 on p. 1826).
2. The associative view being created is the first associative view in the drawing.
3. There are no technical specifications in the drawing.
4. Technical specifications have been entered in the model for which the view is created.

You can perform any actions with respect to transferred specifications that are supported for a drawing's technical specifications: positioning, editing, breaking into pages, deleting (see Section 3.5.7 on p. 1198).

In order to synchronize technical specifications, call one of the following commands in the drawing:

- ▼ **Insertion — Technical Specifications — Synchronize,**
- ▼ **Synchronize Technical Specifications** from the context menu of technical specifications.



If, when creating an associative view, technical specifications are not transferred to the drawing due to non-compliance with one or more above-mentioned conditions, you can transfer specifications later by calling the synchronization command.

When transferring or synchronizing technical specifications, the following peculiarities should be taken into account:

- ▼ text format applied to technical specifications in the model will not be transferred to the drawing;
- ▼ technical specifications are transferred from the model to which the title block is related;
- ▼ previously transferred technical specifications, both edited or entered manually, will be fully replaced by the new specifications.



You can undo a completed transfer or synchronization of technical specifications using the Undo command which is called from the **Standard** bar or from the **Editor** menu.

It should be noted that transferred technical specifications may contain link to different objects created in the model, for example, layout elements – leader lines, base designations, positions, and others.

The links will be active in the transferred technical specifications, if linked objects are present in the drawing in a visible or hidden form. See Section 3.6.3.5.1 on p. 1226 for more information about transferring objects from model to drawing.

3.6.3.10. Copying layers from a model to a drawing

You can copy the layers in the model to the associated view of this model. Each layer contains projections of the objects located on the corresponding layer of the model. Aside from the layers themselves, layer groups and layer filters are also copied if they were created in the model. For details on layers in a model, see Section 2.15.13 on p. 845.

To copy a layer from a model to an associated drawing, perform the following:

1. Enable the projection of a model layer into a drawing. To do this, set the **Project into drawing** attribute to **Allowed**. This is the default value for all manually created layers in a model. You can view or change the layer statuses using the **Document manager** (see 2.15.13.2 on p. 846).



2. Enable the **Layers** switch on the Properties panel when creating a standard or arbitrary view (see Sections 3.6.2.1 on p. 1216 and 3.6.2.2 on p. 1218).



The **Layers** is also available when you create the **In arrow direction**, **Cut/section** and **Projection** views. Its default value in these commands matches the status when the base view is being created. If necessary, you can change the value of the **Layers** switch.



If the projection is disabled for any layer within a mode (the **Project into drawing** attribute is set to **Forbidden**), then the projections of this layer will not be copied to the drawing even if the **Layers** switch is enabled.

The names of the projected layers in the drawing begin with the “(c)” prefix.

When working with the layers projected from a model, keep the following in mind:

- ▼ The layer attributes are copied from a model into a drawing in the following way:
 - ▼ The **Visibility** and **Color** attributes of the model layers perfectly match the corresponding attributes of the drawing layers,
 - ▼ The **Edit** (enabled/disabled) attribute of a model layer corresponds to the **Activity** (active/background) attribute of a drawing layer,
- ▼ You can modify the layer attributes copied from a model in the drawing,
- ▼ You cannot remove or rename the layers copied from a model in the drawing.

You can build new objects on a projective layer. You can copy the projective objects from an original layer to any other layer in the drawing.

Removing a layer from a model automatically removes the corresponding layer from the associated view. Note that:

- ▼ the projections of the model objects on the removed layer are deleted regardless of which layer in the drawing they were located,

- ▼ If a projective layer being removed contains an object from another layer in the model, then this layer will be automatically copied to its initial layer,
- ▼ If a projective layer contains any non-projective objects, then they are automatically copied to the system view of the drawing.

3.6.3.11. Breaking Associated Relations

Associated views permanently maintain a relation with the models that they depict. Due to this, any changes to the model propagates to on its view (views). For the same reason, manual geometry editing in associated views is impossible.

An associated relation between a view and the model can be destroyed. This is done by selecting the **Explode View** command from the context menu of the view in the Drawing Tree. You can also use the **Explode** command from the **Edit** menu.

After exploding, an associated view turns to a set of geometric objects (lines, curves, etc.) and becomes an ordinary user view of a KOMPAS-3D drawing.

The image of an exploded view can be edited by any means available in the system: scaling, deformation, editing using “hot points”, etc. For details on how to edit geometric objects, see Section 3.4 on p. 1113.

When exploding a view which is the base for creating projection views, those views turn into arbitrary ones. In other words, the relation of projective views with the base view is destroyed, while a relation with the model is still maintained.

When exploding a view which is the base for an arrow view and views containing enlarged details or cuts (sections), those views are exploded.

Exploding associated views can be used during the layout stage of laying out drawing development or after finishing working with them, when the relation with the model is no longer relevant. A drawing with exploded (destroyed) associative relations requires less computational resources, it loads and rebuilds faster, etc.



If an associated view was exploded by mistake, press the **Undo** button on the **Standard** bar to restore the previous document state.

3.7. View and fragment insertions. Macro elements

3.7.1. General information on insertions

When working with KOMPAS-3D graphic documents, you can insert previously created images into them. There are several different insertion methods (see Section 3.7.1.3).

- ▼ It is possible to insert images into fragments from other fragments.
- ▼ It is possible to insert into drawings images from fragments or from views of other drawings (for information on types of drawings, see Section 3.5.1.2 on p. 1147).



Fragments and drawings can be inserted also into text documents of KOMPAS-3D. This is detailed in Section 4.1.2.11.8 on p. 1314.

Commands for inserting images are grouped in the **Insert** menu, while buttons for calling commands are grouped on the **Insertions and macro elements** bar (Fig.3.7.1)



Fig. 3.7.1. **Insertions and Macro Elements** bar



An inserted object can be complemented with leader lines which become its part. Such an object is treated by the system (selected, moved, deleted) as a single entity.

The button to call **Object Leader** command is located on the **Insertions and Macro Elements** bar. Detailed information on adding a leader line to an object is provided in Section 3.7.5.3 on p. 1256.

3.7.1.1. Terms and Definitions

Insertion is a type of graphic document object designed for displaying in this document an image from another graphic document or local fragment.

Local fragment is a fragment created and saved in another graphic document.

Container is a document that contains an insertion.

Source of insertion is a document, in which an image used for insertion is created and saved.

View-container is a view of a drawing-container, containing a view insertion from another drawing.

Source view is a type of source drawing, from which an image is inserted into a view-container.

Instance of insertion is an insertion containing an image from a specific source. A document can contain several instances of the same insert, i.e., images inserted from the same view or fragment. Instances can differ in insertion parameters.

Nested insertion is an insertion that is located in the source of this insertion.

3.7.1.2. KOMPAS-3D Fragments

Throughout a professional career, each designer accumulates numerous standard solutions that do not become formalized as a finalized and officially approved drawing. Those can be various tryouts, sketches, drafts, etc. To save them in KOMPAS-3D, a special type of graphic document is designed— the **fragment**. Fragment files have the extension *frw*.

The main difference of a fragment from a drawing is the absence of layout elements, title block, any symbol for the unspecified surface finish and design requirements. Besides that, you cannot create several different views in a fragment. One can say that a fragment is fully similar to the system (reference) drawing view.

If fragment insertion to graphic documents is done often, for the convenience of saving and searching fragments it is possible to use fragment libraries (see Section 12.1.2 on p. 2037).

When working with any graphical document, it is possible to create **local fragments**. These fragments are saved inside graphical document and can be used there many times (see Section 3.7.2.2 on p. 1250).

3.7.1.3. Methods of Insertion

KOMPAS-3D provides three methods to insert views and fragments into other document. These methods are shown in the Table 3.7.1.

Tab. 3.7.1. Methods of fragment and view insertion

Method of Insertion	Description
Embed in Document	<p>Fragment or view contents are copied to the document and saved there as an integral object. In this case, the link to the source fragment is not preserved, except for the information about its name and the complete path to the file.</p> <p>An image embedded in a document can be updated, i. e., acquired anew from the source file. For this purpose, it is necessary that a reference to the source file, including path, be written into the container.</p> <p>When passing a document containing embedded insertions to another workstation, transmission of source file insertions is not needed.</p> <p>Fragments embedded in the document (but not views) can be edited inside this document. Changes will be passed to all insertions of this fragment inside a container, but not to a source fragment.</p>

Tab. 3.7.1. Methods of fragment and view insertion

Method of Insertion	Description
External Reference	In a container, a link to the source file is created without insertion of objects contained in it. Therefore, when editing a source file, all inserts made from that source will be updated. This insertion method is convenient to use when the part or assembly image stored in the source is used simultaneously in multiple different documents, and image modification must lead to an automatic update of those documents (updating of the container takes place upon its activation and opening). Therefore, there is no need to edit each document when a shared part or mechanism is modified. When transmitting the document containing external references to another workstation, it is necessary to pass insertions of source files.
Exploded	Objects located in a fragment or in a view are physically copied into the document, and any link between these objects and the source file is lost.




If the sources of insertions made by external reference are at other workstations, ensure that in the dialog to set access rights to files File Modification Management is enabled. The dialog is called by the **Service — Parameters...** command. — **System— General— Setting access rights** command.








3.7.1.4. Insertion Parameters

After invoking the command for fragment or view insertion, the interface elements appear on the Property bar to manage the insertion. These elements are presented in Table 3.7.2.


Tab. 3.7.2. Controls of fragment insertion management

Element	Description
Source File	Full name of the insertion source file.
 Select other Source of Insertion	Button enabling the indication of another file as the insertion source. After clicking it, the File Selection dialog will appear.
View	Only for view insertion. The list serves to select a source view. It contains names and index numbers of source-drawing views. An image contained in the selected source view is displayed in a ghost view in the document window.

Tab. 3.7.2. Controls of fragment insertion management

Element	Description
	Method of Insertion The group of switches that serves to indicate the fragment or view insertion method (for information on insertion methods see Section 3.7.1.3).
	Insertion Name The field to enter the name of the insertion. This name is displayed in View and Fragment Insertion Manager , and when working with the drawing is displayed also in Drawing Tree and Document Manager . By default, the complete path to the source file is suggested as the insertion name. All instances of the same insertion have the same names. If the name of one insertion instance is changed, it will be assigned to all other instances.
	Mode 1 Only for fragment insertion. The group of switches that controls placing objects from a multilayer fragment.
	<ul style="list-style-type: none"> ▼ If the switch To Current Layer is active, the entire image will be placed in one (current) layer of the fragment or drawing view.
	<ul style="list-style-type: none"> ▼ If the On source layers switch is enabled, the objects of the fragment to be inserted will be located in the fragment or in the view on the layers with the same numbers as in the fragment to be inserted. If there are required layers in the document already, the inserted objects are located on them; layer statuses are not changed (even if they are different from layer statuses in the fragment to be inserted). The required layer will be created with the same status as the respective layers in the fragment to be inserted. Working with layers is described in detail in Section 3.5.6 on p. 1181.
	Extension Lines The group of switches allowing to indicate whether it is necessary to scale the dimensions of the extension lines and leaders in the insertion.
	<ul style="list-style-type: none"> ▼ If the switch Do Not Scale is active, then the length of extension lines and leaders remains the same as in the source file.

Tab. 3.7.2. Controls of fragment insertion management

Element	Description
	<ul style="list-style-type: none"> ▼ If the switch Scale is active, then the length of extension lines and leaders is changed in accordance with current scale of the view-container and/or the scale defined for a fragment insertion. The setting made is saved in the insertion and is accounted for thereafter when changing the scale of the view-container or of the insertion (enabling/disabling of extension lines scaling is ignored when performing these operations).
Base point	The coordinate fields of the insertion's anchor point.
Angle of Rotation	The field of the insertion's rotation angle.
Scale	Only for fragment insertion. The scale field of fragment insertion.
In Thin Lines	<p>Only for view insertion.</p> <p>The option enabling the display of the corresponding thin lines instead of the main and thick lines of the insertion. If it is enabled, then insertion lines are displayed as follows:</p> <ul style="list-style-type: none"> ▼ Main and thick lines, as thin; ▼ Main dashed lines, as dashed; ▼ Main centerlines, as centerlines. <p>The specified replacement is done only for system line styles. User style lines save their appearance despite the status of the option In Thin Lines.</p>
Same as in Document	<p>Only for view insertion.</p> <p>The option enabling the display of insertion objects with default parameters, i.e., parameters defined during setup of the container. If this option is enabled, then display of objects is changed as follows:</p> <ul style="list-style-type: none"> ▼ parameters which cannot be changed during object creation (for example, length of dimension arrows), are set to default ones, ▼ parameters which may be changed during object creation (for example, height of dimension text font) are set to the default ones only if they have not been edited.

3.7.1.5. Insertion Management

The methods of working with insertions such as content editing, parameters editing, source replacement, and deleting can be performed using context menu commands of an insertion selected in a document window (see Sections 3.7.4.1–3.7.4.5). In the drawing, these commands are also available in the context menu of the insertion selected in the Drawing Tree (see Section 3.7.1.7.1).

All above mentioned methods (except for insertion parameters editing) as well as additional features of working with insertions such as renaming, changing of creation method, repeated insertion, local fragments creation and insertion are available in **View and Fragment Insertion Manager**. This is described in Section 3.7.1.6, and the manner of operation for local fragment creation and insertion— in Section 3.7.2.2 on p. 1250.

In **View and Fragment Insertion Manager**, insertions are grouped by types and insertion methods. The operations performed with the insertion are applied simultaneously to all existing instances in the document of this insertion.



Different types of insertions, for which the source is the same drawing, are displayed in **View and Fragment Insertion Manager** as the same insertion.

In the document window, it is possible to select a separate insertion instance and work only with it.

Therefore, **View and Fragment Insertion Manager** is better to use in those cases when it is necessary to change all instances of separate insertion or group of insertions, and the insertion context menu is better to use when the specified instance must be changed.



In the document it is possible also to select several instances of the same or different insertions and work with all selected objects simultaneously.

3.7.1.5.1. Object editing commands applicable to insertions

The following editing commands are applicable to insertions:

- ▼ **Shift** (see Section 3.4.2 on p. 1120),
- ▼ **Rotation** (see Section 3.4.4.1 on p. 1129),
- ▼ **Scaling** (see Section 3.4.4.2 on p. 1130),
- ▼ **Symmetry** (see Section 3.4.4.3 on p. 1131)
- ▼ **Copying** (see Section 3.4.3 on p. 1122).



Note that the result of performing the last two commands is creation of new insertion instances in the document.

It is also possible to move and copy an insertion using the mouse (see Section 3.4.1.1 on p. 1114).

If necessary, the insertion can be converted into a set of individual objects not linked with each other or with the source file. To do that, select an insertion and invoke the command **Edit—Explode**.

3.7.1.5.2. Working with insertion layers

In a container there are all source document objects— and located on visible layers of source and located on the hidden ones. Displaying objects in the main document depends on the status of layers in this document and does not depend on the status of layers in the source document.

- ▼ Controlling the Display of View Insertion Layers is described in Section 3.7.1.7.2 on p. 1247.
- ▼ Controlling the Display of Fragment Insertion Layers is done in the ordinary way— by changing the value of **visibility** property of the respective layer in the container (see Section 3.5.6.5 on p. 1189). Fragment Insertion must be performed on source layers (see Table 3.7.2 on p. 1239).

If you need the change of the object layer visibility not to affect the rest of the image in the document, this layer should not have any other objects except for objects for inserting. The simplest method to reach it: To perform the insertion, assign numbers to layers in the source document that differ from the numbers of layers of the fragment or view, to which the insertion should be made. In this case, as a result of the insertion, new layers for insertion object placement will be created in the main document, and statuses of these layers will be the same as these in the main document.

3.7.1.6. View and Fragment Insertion Manager



To display **View and Fragment Insertion Manager** on the screen (Fig. 3.7.2) click the **View and Fragment Insertion Manager** button on the **Insertions and Macro Elements** bar or select its name from the **Editor** menu.

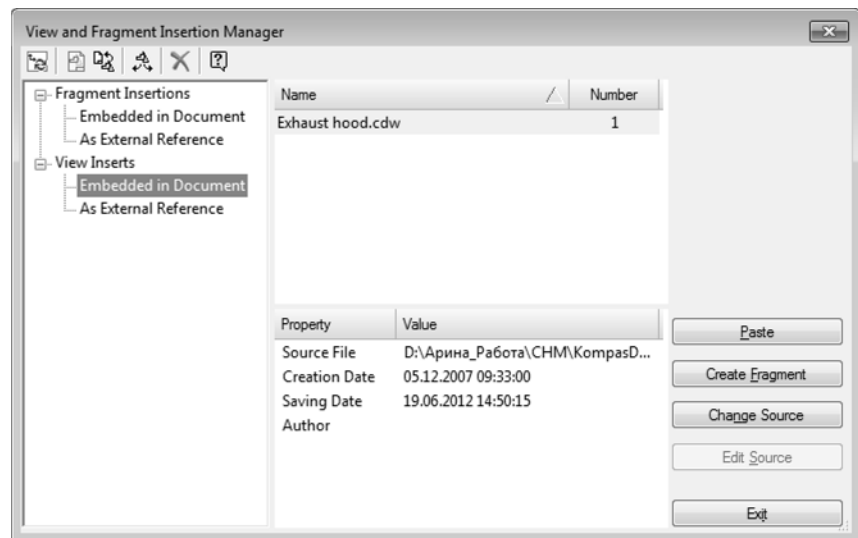


Fig. 3.7.2. View and Fragment Insertion Manager

View and Fragment Insertion Manager serves for the execution of different operations with insertions. Changes you made are applied to all instances of the insertion, on which an operation was performed.



The **View and Fragment Insertion Manager** contains:

- ▼ List of insertion types
- ▼ List of insertions of a selected type
- ▼ List of selected insertion properties




▼ toolbar for insertion control

The description of listed items can be found in the Table 3.7.3.

Tab. 3.7.3. View and Fragment Insertion Manager controls

Element	Description
Insertion Types List	<p>Serves to display the types of insertions found in the document. Depending on the document type and the presence of insertions, it may contain the following sections and subsections:</p> <ul style="list-style-type: none"> ▼ Fragment Insertions <ul style="list-style-type: none"> ▼ Local ▼ Embedded in Document ▼ As External Reference. ▼ View Inserts <ul style="list-style-type: none"> ▼ Embedded in Document ▼ As External Reference.
Insertion List	<p>Serves to display insertions of the type that is selected in the Insertion Types List.</p> <p>This is a table with two columns. In the Name column, names of insertions are displayed, and in the Number column — the number of instances of each insertion. To sort insertions by name or by number, click on the header of the appropriate column.</p> <p>To change the insertion name, click twice on the corresponding cell. When one or several lines in the list are selected, all corresponding instances are selected in the document window.</p>
Properties List	<p>Serves to display the properties of the insertion selected in the List of Insertions.</p> <p>This is a table with two columns. In the Property column, names of properties are listed, and in the Value column — their values.</p>
 Reread the Source Document	<p>Serves for forced updating of an image in insertions embedded in a document. Insertions to be updated must be selected in the List of Insertions.</p> <p>After clicking this button, images in insertions are replaced by those acquired from source files.</p> <p>Updating of an image is possible, if the source file of the insertion is located along the path written in the insertion (this path is displayed as the value of the Source File property).</p>
 Embed in Document	<p>Serves to convert insertions originally included by external reference into insertions embedded in the document. Insertions to be converted shall be selected in the List of Insertions.</p>

Tab. 3.7.3. View and Fragment Insertion Manager controls

Element	Description
	Convert to External Reference Serves to convert insertions embedded in the document, to insertions incorporated by external reference. Insertions to be converted shall be selected in the List of Insertions .
	Explode Used for exploding insertions of any type. Insertions to be converted shall be selected in the List of Insertions . After clicking this button, the link of the insertion to its source file is deleted, and the image is converted into a set of objects; each of them can be edited separately.
	Delete Serves to delete insertions. Insertions to be deleted shall be selected in the List of Insertions . After clicking the button, all instances of selected insertions are deleted from the document. Local fragments and insertions embedded in the document are deleted from the container physically (that is, their content is destroyed). When deleting insertions incorporated by external reference, only the links are deleted, and the source files on the disk remain unchanged.
	Insert Serves to create new instances of the existing insertion. An insertion for which additional instances need to be created must be selected in the List of Insertions . After clicking this button, the View and Fragment Insertion Manager is closed. The process similar to insertion creation is started .
	Create Fragment Serves for creation of a new local fragment in the current graphics document (see Section3.7.2.2 on p. 1250).
	Change Source Serves to change an insertion's source file. Insertions for which the source needs to be changed must be selected in the List of Insertions . After clicking this button, the View and fragment insertion manager is closed and the standard file selection dialog is displayed, in which you need to select the new source for the selected insertions.
	Edit Source Serves for editing insertion content. An insertion to be edited must be selected in the List of Insertions . After clicking this button, the View and Fragment Insertion Manager is closed. The same editing process is started after calling the command Edit Source (see Section3.7.4.1 on p. 1252). Source editing is available for view insertions incorporated by external reference and for all fragment insertions.

* During the insertion creation process started from **View and Fragment Insertion Manager**, it is impossible to change the source file and insertion method.

3.7.1.7. Methods of Working with Insertions in a Drawing

When working with drawings, such navigation tools in the document structure as Drawing Tree and **Document Manager** are available. These tools provide additional capabilities for working with insertions.

3.7.1.7.1. View and Fragments Insertions in the Drawing Tree

In a Drawing Tree (see Section 3.6.1.1 on p. 1207) view and fragment insertions are united in groups **View Insertions (xN)** and **Fragment Insertions (xN1)**, which are dependent on the view that contains the insertions (Fig. 3.7.3). **N** and **N1** in group names are numbers that signify the number of view and fragment insertions respectively.

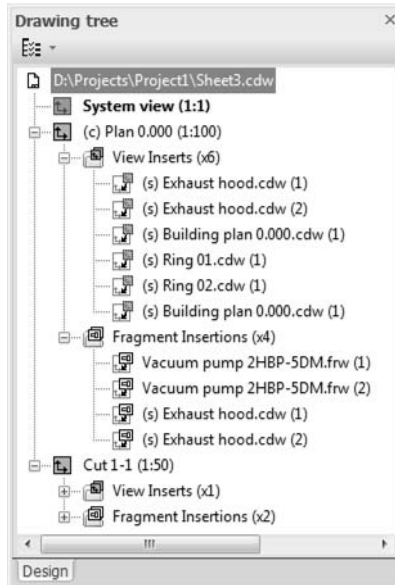


Fig. 3.7.3. Displaying view and fragment insertions in the Drawing Tree

Names of insertions in the Drawing Tree are created by the following template:

X + **name** + (**N**), where

- ▼ **X** is a letter signifying the insertion method:
 - ▼ “L” is insertion of a local fragment,
 - ▼ “S” is an insertion made as an external reference,
 - ▼ No letter means an insertion embedded in a document,
- ▼ **Name** is the name of the insertion;
- ▼ **N** is the index of the insertion instance; the instances are numbered within the view-container; if there is one instance, then the number is not displayed.

When selecting an insertion in the Tree, it is selected in the document window.

The context menu of the selected insertion contains insertion management commands.

3.7.1.7.2. View Insertions in the Document Manager

In the Document Manager (see Section 3.5.6.2 on p. 1182), only view insertions are displayed. In the Tree of Sheets, Views and Layers, they are displayed as subordinate objects of the view-container (Fig. 3.7.4). In the List of Sheets, Views and Layers insertions are displayed after the view-container layer.

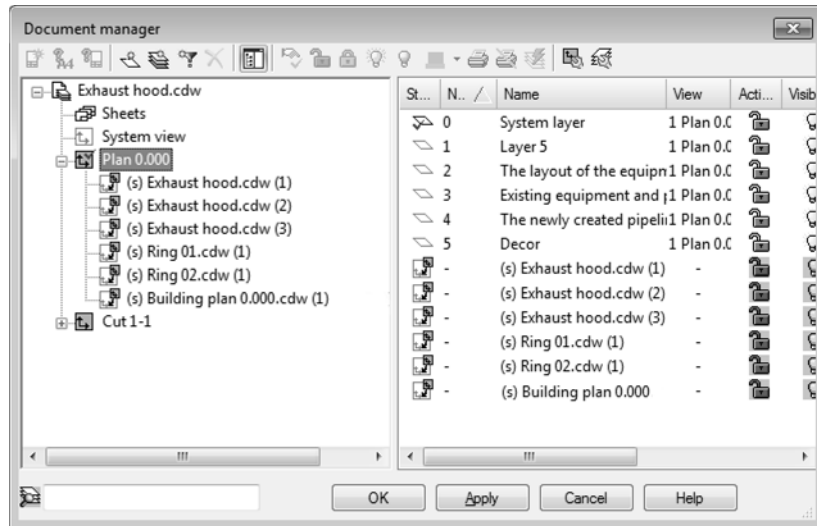


Fig. 3.7.4. Displaying view insertions in the Document Manager

View insertion is always active and visible. Modifying a state is impossible. This insertion peculiarity is marked by special symbols in the **Activity** and **Visibility** columns:



- ▼ impossible to make background,
- ▼ impossible to hide.

It is impossible to make the view insertion current.

You can change the insertion name and comments to it in the corresponding columns of the List of Sheets, Views and Layers.

When selecting a view insertion in the Tree of Sheets, Views and Layers, the layers comprising the insertion are displayed in the List of Sheets, Views and Layers (Fig. 3.7.5).

All insertion layers are always active, and it is impossible to make them background.

None of the insertion layers can be made current.

It is impossible to copy insertion layers to other views, add to groups, or filter.

You can change visibility, and the name and color of insertion layers and comments to them. These changes are not passed to the insertion source view.

Therefore, partial or full disabling of display of insertion objects in the container drawing is possible only through hiding insertion layers. That is why when creating an insertion source view, you should strictly ensure proper distribution of its objects by layers.

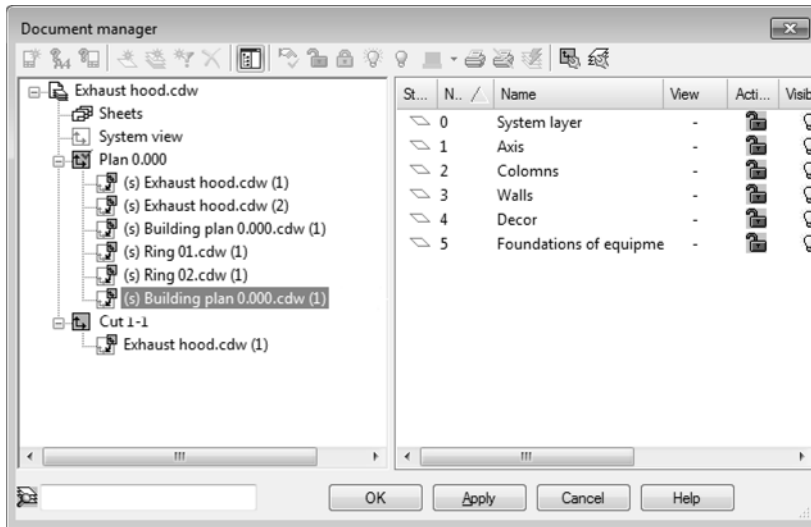


Fig. 3.7.5. Displaying insertion layers in the Document Manager



Layers of nested insertions, i.e., insertions contained in a source view of this insertion, are not displayed in the Document Manager when working with a container drawing.

3.7.2. Fragment insertion

There are two methods of fragment insertion into a graphic document:

- ▼ An insertion of ready fragments from external files (including from fragment libraries, see Section 12.1.2.2 on p. 2039),
- ▼ A creation and insertion of local fragments.

These methods are not mutually exclusive, and they can be combined.

3.7.2.1. Insertion of an External Fragment



To insert an image saved in an external fragment file into the document, invoke the command **Insert Fragment**. To call the command, click the **Insert Fragment** button on the **Insertions and Macro Elements** toolbar or choose its name in the **Insertion** menu.

A dialog will be displayed in which you must specify the fragment path and name to be inserted.

After selecting the file, a ghost view of an image saved in it is displayed, and on the **Options** tab of the Property bar, insertion management controls are displayed (see Tab.). 3.7.2 on p. 1239). Set up insertion parameters in the required manner.

When inserting a parametric fragment, the Property bar also contains the **Variables** tab that enables the administration of external variables values of the fragment being inserted. Parametric fragment insertion is detailed in Section 3.7.2.1.1.



▼ If Auto Create of objects is enabled, then the image is fixed in the document after defining an anchor point position.



▼ If Auto Create is disabled, then to fix an image it is necessary to confirm an object creation.

You can create several insertions of a fragment with different parameters in a single command call.



You can proceed to another fragment insertion without exiting the command. For this purpose, use the **Select Other Source of Insertion** button on the Property bar.

3.7.2.1.1. Parametric Fragment Insertion

A parametric fragment is a fragment that contains external variables. More detailed information on external variables can be found in Section 7.1.5.3 on p. 1593. Let us recall that external variables serve to manage the parameters of the fragment being inserted, without editing it “from inside”.

Modification of external variable values during insertion of a parametric fragment is performed on the **Variables** tab of the Property bar.

Name	Value	C...	Comment
b	58.0	<input type="checkbox"/>	Side B
a	63.0	<input type="checkbox"/>	Side A
c	35.0	<input type="checkbox"/>	Side C
D	12.0	<input type="checkbox"/>	Diameter

Fig. 3.7.6. An example of the **Variables** tab

This tab contains the list of external variables of the fragment being inserted (Fig.3.7.6). Variables have the same values as at the time of last save to disk of the fragment.

To modify the value of a variable, double-click the mouse in the cell, in which it is displayed. Enter the new value and press the *<Enter>* key.

If the variable value is modified, a “checkmark” will appear in the **Changed** column against its name.

To restore a value of some variable to that stored in the source file, click this “checkmark” with the mouse.



The **Table of variables** button serves to open the table of variables of the fragment being inserted and select new values of external variables.

After clicking this button, the **Table of Variables** window is displayed. Select the required row in the table and close the window using the **OK** button. External variables of the fragment will get the values written in cells of the selected row.

The **Table of Variables** button is available if the table of variables saved in the fragment being inserted contains more than one row. The table of variables is built in the file during its creation

or editing, and is saved in it. More detailed information on variable tables can be found in Section 7.1.5.4 on p. 1594.

The changes of variable values are immediately reflected in the ghost view of the fragment in the document window.

After you insert the fragment with certain values of variables, the system waits for new insertion. You can set new values of external variables for it.

If a parametric fragment is embedded in the document or inserted by a link, then external variables and equations associating them with other variables are stored in the insertion itself. Due to this, there can be several fragment insertions with different values of the same variable in one document: variables will not conflict, not “interfering with” each other because each of them controls its own part of the image.

If a parametric fragment is inserted exploded, then immediately after insertion its parameters correspond to the defined values of external variables. But these variables themselves and equations relating them are not passed to the container.

3.7.2.2. Local Fragment Creation and Insertion

To create a local fragment inside the document, do the following.



1. Call the **Create Local Fragment** command. To call the command, click the **Create Local Fragment** button on the **Insertions and Macro Elements** toolbar or choose its name in the context menu by clicking any empty space of the document.
2. Make active and visible all layers (hidden and background) of the current view, or make all views of the current drawing active and visible. The local fragment window opens.
3. Create an image that will be saved in the local fragment.
4. Invoke the command **File— Save— To Container**. The local fragment will be saved in its own main document.
5. In the displayed dialog, enter the name of the created local fragment.
6. Close the local fragment window.



The second method of local fragment creation in the document is clicking the **Create Fragment** button in the **View and Fragment Insertion Manager**.

To insert a local fragment in the document, do the following.



1. Call the **View and Fragment Insertion Manager...** command To call the command, click the **View and Fragment Insertion Manager** button on the **Insertions and Macro Elements** toolbar or choose its name in the **Edit** menu.
2. In **Insertion Types List**, open the section **Fragment Insertions— Local**.
3. In the **List of Insertions**, select the name of the local fragment that has to be inserted into the document.
4. Click the **Paste** button.

Further actions are the same as those during external fragment insertion: define settings on the Property toolbar (See Table 3.7.2 on p. 1239) and specify the base point's position.



When inserting a local fragment, it is impossible to change the source and specify the insertion method.

3.7.3. View insertion

3.7.3.1. Order of Insertion



To insert an image from another drawing view into a drawing, invoke the command **Insert Image from Another Drawing's View**. To call the command, click the **Insert image from another drawing's view** button on the **Insertions and macroelements** toolbar or choose its name in the **Insertion** menu.

A dialog will be displayed in which you need to specify the insertion drawing source path and name.

After selecting the drawing, a ghost view of the image from the source view, i.e., the view that is selected in the **View** list on the Property bar, is displayed.



All layers of the source view are shown in the ghost view, including hidden layers. After creation of the insertion, hidden layers will become invisible.

If necessary, change the source view and set up other insertion parameters using the controls of the Property bar (see Tab. 3.7.2 on p. 1239).



- ▼ If Auto Create of objects is enabled, then the image is fixed in the document after defining an anchor point position.



- ▼ If Auto Create is disabled, then to fix an image it is necessary to confirm an object creation. You can create several insertions of the same view or different views from the source drawing, using a single command call.



You can proceed to insertion of a view from another drawing without exiting the command. For this purpose, use the **Select Other Source of Insertion** button on the Property bar.

View insertion is located in the drawing view that was current during its creation. This view becomes the insertion view-container.

3.7.3.2. Properties of View Insertions

Pay attention to the following special features when working with view insertions.

- ▼ View insertion is not located on any layer of the view-container. It can not be hidden separately from the view-container.
- ▼ In a container, management of insertion layers is available. For this purpose, the **Document Manager** is used (see Section 3.7.1.7.2 on p. 1247).
- ▼ Properties of insertion objects:

- ▼ can be used for construction of other objects (for snapping, as a borders and etc.);
- ▼ They are not available for editing
- ▼ They can not be a source for references
- ▼ They can not be used to create parametric and associative relations.
- ▼ Immediately after creation of an insertion, its scale matches the view-container scale. It may differ from the source view scale and is automatically changed when the view-container scale is being changed.
- ▼ If the source view is associative, its insertion has no direct relation with the model. Hence, if image changes caused by model editing must be passed to a view-container, first the source view must be rebuilt, and then the view-container updated.
- ▼ Items of BOM to which source view objects are related are not added to drawing-container items of BOM.
- ▼ View insertion can be contained in a group.
- ▼ View insertion cannot be contained in a macro element.

3.7.4. Insertion Editing

If an insertion was **exploded**, then inserted objects are not a single entity, i.e., there is no insertion as an object. These objects can be edited separately using any methods, techniques and commands considered in Section 3.4 on p. 1113.

In this section, editing capabilities of insertions made by **external link**, **embedded in document**, and **local fragments** are considered.

3.7.4.1. Insertion Content Editing

Insertion content editing (geometrical objects contained in it) is possible for all types of fragment insertions and view insertions made by external link.

To edit insertion content, select it and invoke the command **Edit Source** from the context menu.

In KOMPAS-3D, a new window will open, in which the external source file, i.e., drawing or fragment (if the insertion was made by an external link), or a set of the insertion's objects (if a fragment was embedded in the document), or a local fragment saved in the document (if a fragment was created directly in the document), will be loaded for editing.

Edit an image in the fragment or insertion's source view and save the document.

If a fragment that is local or embedded in the document is edited, the command **File— Save** contains a submenu, commands of which are shown in the Table3.7.4.

Tab. 3.7.4. Special commands for saving fragments

Command	Description
Save — To Document-Container	This enables saving just an edited fragment with auto update (but without saving) of a container.
Save — With Container to File	This enables saving simultaneously an edited fragment and a document-container. If a document-container is being saved for the first time, the dialog for entering the file name will appear.

After saving the fragment, close its window and return to the document-container. The changes made to the fragment will be immediately reflected in all insertions existing in the main document.

Access to the content of an insertion made by external link is possible without referring to the document-container. The source file of such an insertion can be opened as usual and edited. In this case, the document-container update will be done automatically upon opening, and if it was already opened— upon activation. Hence, the source file and the document-container can not be opened and edited at the same time, but the correspondence between them will always be maintained.

3.7.4.2. Editing Insertion Parameters

Insertion parameters are parameters set on the Property bar during insertion creation (see Tab. 3.7.2 on p. 1239). Some of them can be changed after the insertion creation.

For this purpose, click on the insertion twice, or select it and invoke the **Edit** command from the context menu or from the **Editor** menu. The Property bar will display the same tabs as when creating this insertion.

You can change any insertion parameters except for the path to the source file and the insertion method. For insertion of a parametric fragment, it is also possible to set new values of variables.



To change the path to the source file (actually to change the source), use the command **Change Source** (see Section 3.7.4.3).

To change the insertion method, use the **View and Fragment Insertion Manager** (see Section 3.7.1.6 on p. 1243).

Every change is reflected on the insertion ghost view in the document window. After finishing editing, click the **Create Object** button.

3.7.4.3. Replacing the Source

To replace an insertion source, select the insertion and invoke the **Replace Source...** command from the context menu.

After invoking the command, the Select File dialog appears, where you need to select a document that will be the source of the insertion.

After that, the process of insertion creation will start. Insertion parameters (scale, rotation angle, etc.), for which the source was changed, remain the same. These parameters are displayed in the fields on the Property bar. If necessary, you can change them. Upon finishing the settings, click the **Create Object** button on the Advanced Control panel.



In the same way, you can change the source for several insertions, i.e., replace all selected fragment or view insertions (even if insertions' source files are different) by any one fragment or view. For this purpose, select in the document those insertions, for which the source must be changed, and select the context menu command **Replace Source...— Of View Insertions/Of Fragment Insertions**.

After specifying the new source file, it will be embedded in the document in place of each of the selected insertions, while saving their parameters. To confirm the insertion source change, click the **Create Object** button.

3.7.4.4. Updating Insertions

The insertion update is transfer of changes from the insertion source to the container.

The changes in the insertion source are required to be saved to update insertions.

- ▼ If the insertion is made by an external link, it is updated automatically when opening or activating the container.
- ▼ If the insertion is embedded in a document, the source is required to be reread to update it. This can be done with the **Reread** button in the **View and Fragment Insertion Manager** (see Table 3.7.3 on p. 1244).



3.7.4.5. Deleting Insertions

If any of the insertions are no longer needed, you can delete them from the document. For this purpose, select the insertions in the document window and press the *<Delete>* key.

Since a local fragment is saved inside the document, it remains there even when deleting all of its insertions. Thereafter, a local fragment can be inserted again (see Section 3.7.2.2 on p. 1250).

If it is necessary to delete from the document not a local fragment insertion but the local fragment itself, use the **View and Fragment Insertion Manager** (see Section 3.7.1.6 on p. 1243). Using it, you can also delete any other insertion, including one that is not displayed in the document, for example, a view insertion, all layers of which are hidden or an insertion made by external link, the source of which is not found.

3.7.5. Macroelements

When developing products, the designer uses a large number of standard parts and mechanisms. Those include various fastener parts (bolts, nuts, screws, washers), bearings, switches, jacks, and so on.

When editing a drawing, you may need to highlight, move, rotate such parts as a whole, since actions to each object separately will take much time and will not be convenient. In addition, some image elements of a standard part must not be accessible for editing.

For this purpose, you can put together different objects of the KOMPAS-3D image — geometric objects, dimensions, hatches, designations, and others — into **macro elements**.

Objects merged into macroelement must be located on one drawing view (for more on views see Section 3.5.4 on p. 1160).



Associative view objects maintaining relation with the models cannot be included into the macro element.

Objects as part of a macro element cannot be highlighted, deleted, or edited separately (except for changing style or a leader created in it). Thus, a macro element is processed by the system as a single entity.

You can gain an access to individual objects belonging to a macro element after exploding or when editing it. In this case, no relation is maintained between the objects that used to belong to the macro element. When editing a macro element, you can perform different actions to its components without link interrupting the link, and add new objects to the macro element.

Drawings of standardized parts inserted from the KOMPAS-3D libraries are macro elements. You can create your own macro element application libraries. Also you can customize macro element properties (mass, power, cost per meter, etc.). KOMPAS-3D reports can be formed automatically using the properties of macro elements inserted into a drawing. Setting of properties is described in Section 5.1 on p. 1369, report formation— in Section 5.2 on p. 1403.

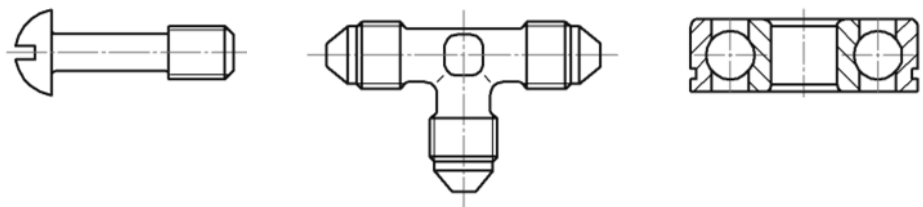


Fig. 3.7.7. Samples of macro elements from a library

The buttons to call the commands to work with macro elements are grouped on the **Insertions and Macro Elements** bar (see Fig.3.7.8). More information about macro element creation and working with them is provided in Sections 3.7.5.1–3.7.5.8.



Fig. 3.7.8. **Insertions and Macro Elements** bar



On the **Insertions and Macro Elements** bar the buttons to call the view and fragment insertion commands are located. Insertions are described in Sections 3.7.1 – 3.7.3.

3.7.5.1. Creating macro element

To join objects into a macro element, complete the following steps.

1. Select all the objects which must be included into a macroelement (the selection of objects is described in Section 3.1.3 on p. 876).
2. Call the **Create Macro Element** command. To call the command, click the **Create Macro Element** button on the **Insertions and Macro Elements** toolbar or choose its name in the **Service** menu.



If objects from different views are selected, the command will be unavailable.

If the objects selected for inclusion into a macro element are located on different layers, the screen will display a dialog requesting to specify the macro element type: multilayer or single-layer type.

A macro element icon appears in the Drawing Tree (see Fig.3.7.12 on p. 1261).

3.7.5.2. Manipulations with macro elements

The macro elements, just like the individual objects, can be subjected to various actions: copying to the buffer, exchange, deleting, moving, rotating, etc. A macro element is treated by the system as a single entity.

Before performing an operation, you need to select a macro element. To do this, click it with the mouse or select it in the Drawing Tree. You can simultaneously select several macro elements in the document window.

To select all the macro elements in a current drawing or a fragment at once, call the **Select—By Type** command, and then select **Macro Elements** in a list of existing objects.

After selecting a macro element, call the desired command, for example, from **Editor** menu and perform actions provided in the command.

3.7.5.3. Object Leader

You can create and add to an object one or several leader lines which become its part (see Fig.3.7.9).

Leader line creation and adding is performed with **Object Leader** command.

Objects to which a leader line can be added are:

- ▼ macro elements (including library macro elements),
- ▼ fragment insertions,
- ▼ image inserts from views of other drawings.

An object with such a leader is treated by the system (selected, moved, deleted) as a single entity. Therewith, the leader lines can be edited separately from its initial object.

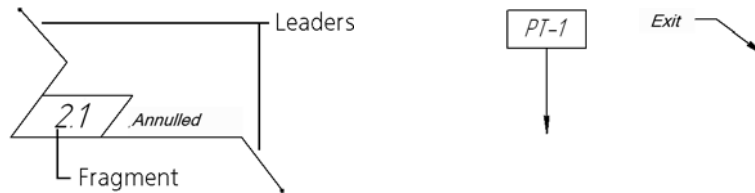


Fig. 3.7.9. Samples of objects with leader lines

3.7.5.3.1. Leader Creation

To create or add a leader line to an object, do the following,

1. Select one object— macro element or insertion to which a leader line must be added.
2. Call the **Object Leader** command

To call the command:



- ▼ by clicking the **Object Leader** button on the Insertions and **Macro Elements** bar;
- ▼ from the **Tools** menu;
- ▼ from the object context menu in the document window or in the Drawing Tree.

For macro element the command is also accessible in the Macro Element Editing mode (for more about editing mode see 3.7.5.4 on p. 1258).

3. Customize the start point of the first branching.

Set the point of joining the leader to the object **t1**.



Point **p1** is related to the object: hereafter, when the object is edited, the point is displayed as one of the handles which, if moved, moves the whole object. In some cases, it may be more convenient to align the point of joining the leader with the point of moving the object (see Fig. 3.7.10). At the same time, some library macro elements can have defining points for dimension control. Joining of a leader with one of such points is not advisable, since it will lose its function.

Then, select the start points of other branchings. Their number is not limited.

4. Set up branchings display. Specify parameters in the same way as for usual leader line (see Section 3.3.7.4.2 on p. 1025). Note that when adding leader line to an object, shelf and caption control elements are unavailable on the Properties bar.

The ghost of the created Leader is displayed. You can edit or delete any of its branches without leaving the command dialog (see Section 3.3.7.4.3 on p. 1027).



To lock the image, click the **Create Object** button.

To add to the object next leader line, call the **Object Leader** command again.

3.7.5.3.2. Objects editing and leader line deleting

Editing of objects with leader lines

When highlighting an objects with a leader, the objects hot points and leaders are displayed on it (Fig.3.7.10). You can edit the object using points for movement and rotation and the point of the leader connection remains bound to the object. The leader “behaves” the same way as a normal leader when moving a shelf’s starting point: it becomes longer, shorter, or rotates.

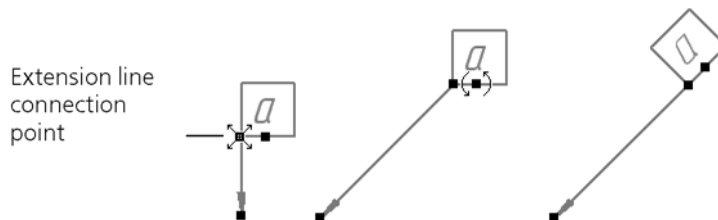


Fig. 3.7.10. Editing of objects with leader lines sample
(connection point of the leader to the objects coincides with its movement point)

Editing of leader line points is performed by using following methods.

- ▼ To change branchings position, select an object and move its defining points (see Section 3.4.1.1.3 on p. 1114).
- ▼ To change configuration of a leader, double-click it with a mouse and perform actions related to adding and deleting points or branches (see Section 3.3.7.4.3 on p. 1027).



When exploding a macro element, the leader line contained in it is transformed into the usual leader line.

Deleting Leaders

To delete leader line of an object, select it in the document window or in the Document Tree. Then call the context menu **Delete Leaders** command. The command appears in the menu if one object with designation created by the **Object Leader** command is selected.

After the command activation, all the leader lines of object are removed.

Deleting one leader from a series of them is only possible for a macro element. To do this, go into its editing mode, select a leader line to be deleted and press the <Delete> key.

3.7.5.4. Macro element editing mode

Macroelement editing is carried out in the editing mode.

The following actions can be performed in this mode:

- ▼ Edit individual objects of the macro element.
- ▼ create new or add existing objects to macro element,
- ▼ specify and delete macro element defining point,
- ▼ add and delete macro element leaders (see Section 3.7.5.3 on p. 1256).



To switch to the macro element editing mode, you need to double click the macro element or select it and call the **Edit Macro Element** command. To call the command, click the **Edit Macro Element** button on the **Insertions and Macro Elements** or on the **Current Status** toolbar, or select it by the name from the **Editor** menu.

While the system is in the macro element editing mode, the **Edit Macro Element** button stays pressed. The mode symbol appears in the document window. Macro element being edited is marked with a “lock” icon in the Drawing Tree (Fig.3.7.12, b).

Edit the macro element objects. Any other objects are unavailable—they are displayed on the screen in the background view display style.

Create new objects (including multilayer) or delete the existing. All objects constructed in the macro element editing mode are automatically included into its contents.



To add an object from among those present in the document, use the **Add geometry** command. For information about working with the command, see Section 3.7.5.5 on p. 1259.



To create or delete macro element handle, use the **Specify Handle** command. See Section 3.7.5.6 on p. 1260 for more information about operation of this command. Editing a macroelement's point is described in Section 3.7.5.6.1 on p. 1260.



To create a leader line in the macro element, use the **Object Leader** command. For information about working with the command, see Section 3.7.5.3.1 on p. 1257.

To exit editing mode, release the **Edit macroelement** button or select the command with similar name from the **Editor** menu again or click the edit mode symbol in the document window.

3.7.5.5. Adding Objects to a Macro Element

Add Geometry command allows to add objects of document to macro element.

To add objects to a macro element, complete the following steps.

1. Select one macro element to which objects need to be added or enter its editing mode.



2. Call the **Add Geometry** command. To call the command, click the **Add Geometry** button on the **Insertions and Macro Elements** toolbar or choose its name in the **Tools – Macro Elements** menu.

3. Select objects to be added by clicking them with mouse.

When an object falls into the cursor trap, the object is highlighted.



Objects located in another view are inaccessible for selecting.



To ensure that the source objects will be automatically deleted, activate the **Delete Source Objects** switch. The activation of the switch **Retain Source Objects** means that they will be saved.



To finalize the command, pres the **Interrupt Command** button.



3.7.5.6. Setting handling points

The **Set Handle** command allows to create a hot point in a macro element and delete a previously created hot point. The handle, if present in the macro element, is used to move or rotate it when editing the image.

To set a defining point, complete the following steps.

1. Select one macro element to which the defining point needs to be added or enter its editing mode.
2. Call the **Specify Handle** command. To call the command, click the **Set Handle** button on the **Insertions and Macro Elements** toolbar or choose its name in the **Tools — Macro Elements** menu.
3. Specify position of the reference point with the mouse or enter its coordinates on the Properties menu. To do this, you can use snaps or geometrical calculator.

After selecting, the symbolic notation of the point appears on the screen.

To delete previously created defining point, upon calling the command click the **Delete Handle** button on the Special Control panel. The button is active if this point was created in the macro element.

To finalize the command, pres the **Interrupt Command** button.



3.7.5.6.1. Changing handle's position with respect to the macro element

After setting a handle, you can move it with respect to the macro element by setting its new position.

For this purpose, select the macro element in which the point is located or enter its editing mode. Call the **Specify Handle** command. Then, remove fixation of coordinates on the Property bar and specify new position of the point.

3.7.5.6.2. Using a macro element handle

Hot points are used to position a macro element in the document using snaps.

The hot point is a movement point and the rotation center for a macro element. There is another point near it which is used to change the macro element's slope angle (see Fig.3.7.11). You can rotate macro element with mouse by moving the second point. For exact turning, you can match this point with another object using a snap.



Fig. 3.7.11. Handle Display: a) in the macro element editing mode, b) when editing a document

3.7.5.7. Displaying macro elements in the Drawing Tree

In a Drawing Tree (see Section 3.6.1.1 on p. 1207) macro elements are united in groups **Macro (xN)**, which are dependent on the view that contains the macro elements (Fig.3.7.12, a).

N in group names— numbers that signify the number of macro elements in each of them.

When creating or inserting, the macro element is given a default name and number which can be edited in the drawing's Tree.

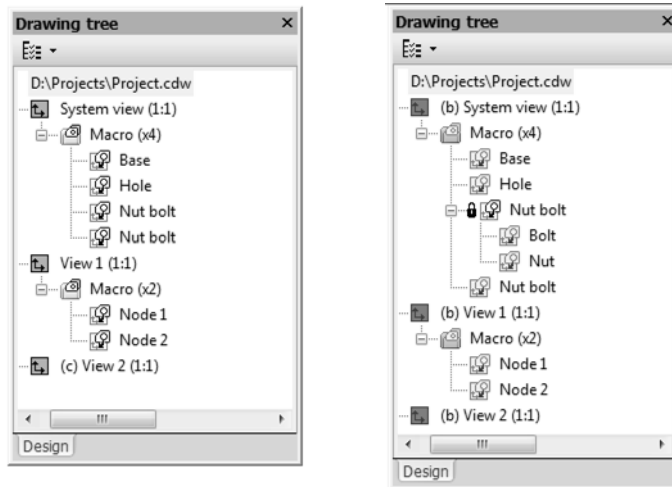


Fig. 3.7.12. Displaying macro elements in the Drawing Tree:
a) in the construction mode; b) in the object editing mode

When selecting an insertion in the Drawing Tree, a macro element is selected in the document window.

Context menu of the selected macro element contains commands to control macro element (Fig.3.7.13).

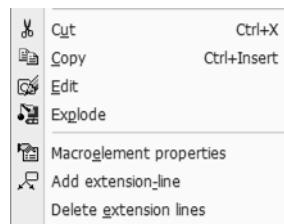


Fig. 3.7.13. Macro element context menu in the Drawing Tree

When editing macro element that consists of other macro elements, its structure is displaying in the Drawing Tree. In this case, the “+” icon is displayed to the left of macro element name in the Tree. Click the icon to display the contents of macro element (Fig.3.7.12, b).

3.7.5.8. Exploding a Macro Element

You can explode a macro element (including one which is inserted from a library) into separate component objects.

To explode a macro element, complete the following steps.

1. Select a macro element (macro elements), which must be exploded.
2. From the menu, call the **Explode** command from the **Edit** menu.

After exploding, the macro element is automatically deleted from the Drawing Tree.

3.8. Measuring in Graphic Documents

3.8.1. General Information

When working with graphical documents you may need to find out the distance or angle between points, curves, the area of shapes, etc. documents supports measurements of various geometrical characteristics, as well as calculation of mass-inertia properties of a body of revolution or an extrusion (volume, mass, center of gravity coordinates, axial and centrifugal moments of inertia).

Measuring commands are grouped in menu **Service**, and command buttons are on the **Measurements (2D)** toolbar (Fig. 3.8.1).

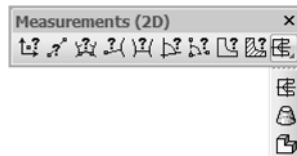


Fig. 3.8.1. 2D Measurements Panel

3.8.1.1. Quick view of measurement results

You can quickly view measurement results or enter them into the Information window.

To view measurement results, move the cursor to an object to be measured (e.g. to the curve with its length to be measured) after calling the command. Measurement results will be displayed next to the cursor (Fig. 3.8.2, a).

If several objects are involved in the measurement, select all of them by clicking them one by one except for the last one. Move the cursor to this object. Measurement results will be displayed next to the cursor. For instance, you need to measure distance between two points on a curve. Click the curve along which the distance will be measured. Then click the first curve point and move the cursor to the second curve point. Measurement result will appear next to the cursor (Fig.3.8.2, b).

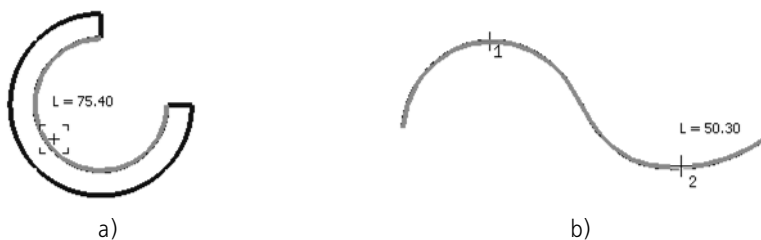


Fig. 3.8.2. View of measurement results:
a) curve length; b) distance between two points on a curve

To enter measurement results into the Information window, you should click a measurement object (or all measurement objects, if many). The Information window will pop up. The window displays measurement results and some info parameters. Window management and its menu commands are described in Section 3.8.1.2.

3.8.1.2. Information Window

The Information window pops up after you click measurement objects.

For example, Figure 3.8.3 shows the Information window when measuring the distance from the curve to the point.

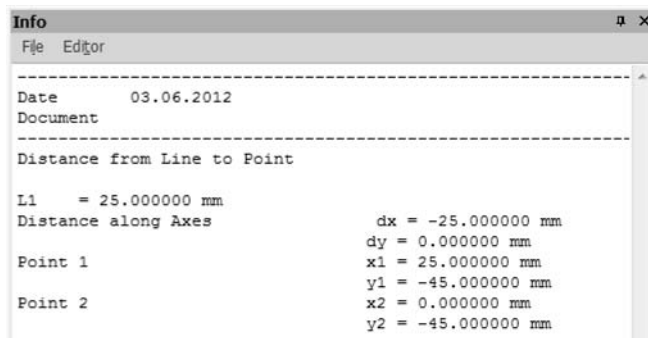


Fig. 3.8.3. Example Information Window

It shows:

- ▼ The current date,
- ▼ full name of the active document,
- ▼ name of the current command,
- ▼ measurement results.

You can move the cursor through the text in the window, enter your own text, and select text fragments with keyboard or mouse. If the length of the results list exceeds window size, you can use scroll bar for review.

The information window has its own menu consisting of two items: **File** and **Editor**. Menu commands are shown in the Tables 3.8.1 and 3.8.2 respectively.

Tab. 3.8.1. **File** menu commands

Command	Function
Save	To save Information window contents in text file* (*.txt). Upon calling the command, a dialog will appear. Enter the file name for saving**.
Printing...	To print Information window contents. Upon calling the command, the Print Settings dialog will appear.
Exit	To finish the current measuring command without saving results.

- * File is saved in Windows text format and may be used later in any Windows applications.
- ** If the existing text file is selected, the inquiry “*Add text at the end of file?*” appears on the screen. If the answer is affirmative, text measuring results will be added to file text; otherwise, the text will be replaced.

Tab. 3.8.2. **Editor** menu commands

Command	Function
Cut	To cut selected text fragment and copy it to the Windows Clipboard.
Copy	To copy selected text fragment to the Windows Clipboard.
Insert	To paste text from the Windows clipboard.
Delete	To delete selected text fragment.
Select All	To select entire text in the Information window.
Font...	To select text font in the Information window. Upon calling the command, font setup dialog will appear.
Find...	To find selected text fragment. Upon calling the command, the dialog for entering text or set up search parameters will appear on the screen.



Information in the Information window is updated after setting a new measuring object. Therefore, editing text in a window is only recommended after completing measurements.



Fixation button in the Information window heading enables its automatic minimizing mode.



Close button allows to finish the current measuring command.



3.8.2. 2D Measuring

After calling any 2D measuring command (see Fig.3.8.1 on p. 1263), **Measuring** tab appears on Property toolbar. It contains elements allowing to customize measurement process parameters. These elements are presented in Table 3.8.3.

Tab. 3.8.3. Control elements of measurement parameters

Element	Description
Center, p1, p2, p	Point coordinate fields set for measuring.
Distance	Information field showing the measured distance.
dX, dY	Information fields showing the distance between set points on in current coordinate axes. Only for the Distance between Two Points command.

Tab. 3.8.3. Control elements of measurement parameters

Element	Description
Angle	Information field showing the measured angle (for Angle between Two Lines and Angle by Three Points commands) or slope angle of radius vector of set point to the current OX coordinate axis (for Point Coordinates command).
Decimal Places	The list controlling representation of measurement results. Minimal decimal place is 0, and maximum is 10.
Length Units	The list allows selection of length units: millimeters, centimeters, meters.
Angular Units	The list allowing selection of angle units: degrees, radians.
Mode	This switch allows to select the length and area measurement mode. It appears on the Property toolbar if the scale of the current view is other than One. Activating Do not Consider View Scale switch allows to obtain the “full-size” result. Activating On Paper switch allows to obtain the result in view scale ^{**} .
Point/ Points*	Group of switches controlling auxiliary points drawing formed in the measuring process. Set of switches in this group depends on selected measuring command (see Table 3.8.4).
 Center of Gravity*	Group of switches controlling point drawing in the center of mass. Only for the Area command.
 Style*	The list allows selection of the point style.
Bisector*	Group of switches controlling the drawing of measuring angle bisector. Set of switches in this group depends on selected measuring command (see Table 3.8.5).











* Action of control elements **Points**, **Mass Center**, **Style** and **Bisector** is extended to the following measuring.

** Linear values measured “on paper” are view scale values multiplied by “full-size” values, and area values are “full-size” values multiplied by square of scale.

Tab. 3.8.4. Content of the group of switches **Point/Points** depending on the active command

Command	Content of Point/Points group
Point Coordinates	





Tab. 3.8.4. Content of the group of switches **Point/Points** depending on the active command

Command	Content of Point/Points group
	Do Not Draw Measured Point
	Draw Measured Point
Distance between Two Points	
	Do Not Draw Midpoint
	Draw Midpoint
Distance between Points On Line	
	Do Not Draw Measured Points
	Draw Measured Points
Distance from Curve to Point	
	Do Not Draw Measured Points
	Draw Measured Point
Distance between Two Curves	
	Do Not Draw Nearest Points
	Draw Nearest Points

Tab. 3.8.5. Content of the group of switches **Bisector** depending on the active command

Command	Content of Point/Points group
Angle between Two Lines/Segments	

Tab. 3.8.5. Content of the group of switches **Bisector** depending on the active command

Command	Content of Point/Points group
	Do Not Draw Bisector
	Draw Bisector
Angle by Three Points	
	Do Not Draw Bisector
	Draw Bisector

3.8.2.1. Point Coordinates



To define location of an arbitrary point in graphical document, call the **Point Coordinates** command.

Specify points whose coordinates are to be defined.

You can quickly view measurement results by moving the cursor to a point or enter the measurement results into the Information window by clicking the point (for the quick view see Section 3.8.1.1 on p. 1263).

During quick view, point coordinates in the current coordinate system are displayed next to the cursor.

The Information window displays the following measured values:

- ▼ point coordinates in the current coordinate system,
- ▼ distance from coordinate system origin to the point,
- ▼ angle between radius vector of the point and X axis.

The distance and angle are polar coordinates of the point in a system whose axis coincides with X axis of the current Cartesian coordinate system.

3.8.2.2. Distance between Two Points



To define the distance between two arbitrary points of the graphical document, call the **Distance between Two Points** command.

Specify two points to measure the distance between them.

The system will determine the shortest distance between the selected points (length of the line segment joining them).

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the distance value between the selected points is displayed next to the cursor. In the Information window the distance between points is supplied.

mented with: the distance between points along axes of the current coordinate system and some info parameters.

3.8.2.3. Distance between Points on Curve



To measure the distance of curved line limited by two points, call the **Distance between Two Points on Curve** command.

Select a curve to measure the distance between 2 points on it.

Then set up two points to limit measured line.

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the length value of the curve segment bounded by the selected points is displayed next to the cursor. In the Information window the curve segment length is supplemented with some info parameters.



If the specified points do not belong to the selected curve, the region boundaries position will be determined by the projection (mapping) of the specified points on the line. Use snaps for precise positioning of the cursor (see Section 3.1.1 on p. 865).

If the close curve is selected, the measured line should also be indicated.

3.8.2.4. Distance from Curve to Point



To measure the distance between a certain curve and an arbitrary point, call the **Distance from Curve to Point** command.

Select the curve, and then set up the point, distance to which is to be measured.

The system will define the length of the shortest perpendicular dropped from the point to the curve.

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the distance value from the curve to the selected point is displayed next to the cursor. In the Information window the distance value is supplemented with some info parameters.

3.8.2.5. Distance between Two Curves



To define the distance between two curves, call the **Distance between Two Curves** command.

Select two curves with cursor.

The system will define the shortest distance between them.



Only non-zero distance measuring is available. Therefore, after selecting the first curve, you may set up only a curve non-intersecting to it.

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the distance value between the selected curves is displayed next to the cursor. In the Information window the distance value is supplemented with some info parameters.



The **Measure Distance between Curves** command may be used to define the nearest points of two elements.

3.8.2.6. Angle between Two Lines/Segments



To define the angle between straight objects, call the **Angle between Two Lines/Segments** command.

Select objects for measuring angles between them. Such objects may be lines, segments, segments of polylines, and sides of polygons in different combinations.

The angle is dimensioned, i.e., created by a point on the first object nearest to the location of object selection, the intersection point of the objects or their extensions (projections), and the point on the second object nearest to the location of selecting the latter object.



If parallel objects are selected, the angle value will be zero.

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the angle value between the selected straight objects is displayed next to the cursor. In the Information window the angle value is supplemented with the difference between the measured angle and 180° and 360° angles.

3.8.2.7. Angle by Three Points



To measure the angle formed by three arbitrary points, call the **Angle by Three Points** command.

Set up the angle vertex and two points on its sides.

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the angle value between the selected points is displayed next to the cursor. In the Information window the angle value is supplemented with the difference between the measured angle and 180° and 360° angles.

3.8.2.8. Curve Length



To measure the full line length (i.e., length of open curve from the starting point to end point or close curve perimeter), call the command **Curve Length**.

Select the curve which length is to be measured.



For length measuring, only lines located in the current view (see Section 3.5.5.2 on p. 1171 for information on changing view states) may be selected.

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the length value of the selected curve is displayed next to the cursor. The same value is displayed in the Information Window.

You can view length values of other curves by moving the cursor to them. To enter the measurement results into the Message window, click these curves one by one.

The selected objects are highlighted. The Information window will display the list of measured lengths. The sum of the measured values will appear at the end of the list.

Select the line again to exclude it from the list. Selection of this line will be canceled, its length measurement will be deleted from the window, and the length sum will be calculated again.



If the line contains portions of some intersecting curves, select it as passing by arrow. To do this, click the **Border by Arrow Direction** button. The further sequence of operations is described in Section 3.2.13.1.2 on p. 957. After the contour is generated, the measurement result will appear in the Information Window.



If a polyline which does not coincide with any available contours is to be measured, it should be created manually. To do it, click the **Manual Drawing of Borders** button. The further sequence of operations is described in Section 3.2.13.1.1 on p. 956. After the contour is generated, the measurement result will appear in the Information Window.

3.8.2.9. Area



To define the area of arbitrary shape, call the **Area** command.

Select the point inside closed area limited by intersecting geometric objects. The borders of the shape created by these objects will be defined automatically. The measured area will be highlighted in a certain color.



For area calculation, you may use only those objects which are located in the current view (see Section 3.5.5.2 on p. 1171 for information on changing view states). T

You can quickly view measurement results or enter them into the Message window (see Section 3.8.1.1 on p. 1263).

During quick view of measurement results, the area value of the selected shape is displayed next to the cursor. The same value is displayed in the Information Window.

You can view area values of other shapes by moving the cursor to them. To enter the measurement results into the Message window, click inside these shapes one by one.

The Information window will display the list of measured areas. The sum of the measured values will appear at the end of the list.

If already measured area is selected again by the mouse, it will be deleted from the list of measured areas in the Information window. The list will be re-numbered, the sum of all measured areas will be recalculated, and color highlighting will be canceled.

Using the Special Control Panel buttons, you can calculate the area of:



- ▼ A figure limited by a closed contour (circle, ellipse, polygon, etc.). For this purpose, click the Specify closed curve and **specify the required contour**.



- ▼ Figures formed by a set of geometric objects. To do this, click the Border by **Arrow Direction** button. The further sequence of operations is described in Section 3.2.13.1.2 on p. 957. After the contour is generated, the measurement result will appear in the Information Window.



- ▼ Figure, the borders of which do not exist in a drawing. To do it, click the **Manual Drawing of Borders** button. The further sequence of operations is described in Section 3.2.13.1.1 on p. 956. Temporary polyline will be formed. After the contour is generated, the measurement result will appear in the Information Window.



To measure the area and calculate the geometric center of a complex shape containing holes and several parts, use **Calculate Mass-Inertia Properties of 2D Body** command (see Section 3.8.3.2 on p. 1273).

3.8.3. Mass-Inertia Properties

The commands for MP calculation are united into one group on the **Measurements (2D)** toolbar (see Fig. 3.8.1 on p. 1263).

Upon calling any MP calculation command, **Measure** tab appears on the Property bar. It contains elements allowing to set up the calculation process parameters. These elements are presented in Table 3.8.6.

Tab. 3.8.6. Control elements of MP calculation parameters

Element	Description
p	These are the coordinate fields of the coordinate system origin, with respect to which the Mass-Inertia properties of the body will be calculated. The point coordinates are set up in the current coordinate system.
Angle	These are the rotation angle fields of the coordinate system, with respect to which the Mass-Inertia properties of the body will be calculated. This angle is counted from OX current coordinate axis.
Number of Digits	The number of significant figures is the number of figures in decimals without considering zeroes in their beginning. In case of an exponential record, the number of significant figures is the total number of signs in integral and decimal parts. The minimum number of digits is 1, the maximum is 10.
Length Units	The list allows selection of length units: millimeters, centimeters, meters.
Mass Units	The list allows selection of mass units: grams, kilograms.

Tab. 3.8.6. Control elements of MP calculation parameters

Element	Description
Center of Gravity	Clicking this button allows the drawing of point in the center of gravity (for 2D shapes) or point projection of a body center of gravity to the drawing plane (for revolved or extruded bodies).
Style	The list allows selection of the point style.

3.8.3.1. Object Borders Setting up

To calculate MP of a 2D shape, it is necessary to set up its borders; to calculate MP of revolved or extruded bodies, set up body section borders. There are several methods to set up borders.

- ▼ If the document contains close contours (ellipses, circles, splines, etc.) limiting the shape or body section, you need to specify them. This method is default, i.e., upon calling the MP calculation command, system is pending for contour selection. The selected border may be excluded from calculation group. To do that, select it once again.



- ▼ If you need to select contour (created by set of geometric objects) as a border, click the **Border by Arrow Direction** button. The further sequence of operations is described in Section 3.2.13.1.2 on p. 957. If you followed the direction arrow and created an open contour, its closing line will be created automatically.



- ▼ If there is no shape or body section in the drawing, you may create temporary polyline. To do it, click the **Manual Drawing of Borders** button. The further sequence of operations is described in Section 3.2.13.1.1 on p. 956.

After selecting each contour, **Object Properties** dialog appears on the screen. Specify contour limitation as **body** or **hole**.

Each new contour is added to selected contours, and calculation is performed for complex shape (complex body) until MP calculation command execution is finished. To start a new calculation, finish command execution and call the command again.

3.8.3.2. MP of 2D Shapes



To calculate area, center of gravity coordinates, axial and centrifugal angular momenta of a 2D shape, call the **MP Calculation of 2D Shapes** command.

Set up borders of the shape and its holes.

MP calculation begins right after the first border is selected. When a new border (of the hole of body) is selected, MP are recalculated.

3.8.3.3. MP of Revolved Bodies



To calculate MP of a revolved body (or its sector) with selected generating section, call the **MP of Revolved Bodies** command.

Revolved body for MP calculation is generated according to the following rules.

- ▼ The generating section lies in the XOY plane (in the plane of the drawing) on one side of the X axis.
- ▼ Section revolves about the X axis.
- ▼ The angle of revolution arc for revolved body is 360°. For revolved body sections, the 'Included Angle' of the revolution arc is less than 360 degrees.

Set up borders of body as XOY plane.

Upon selection of each contour, object properties dialog appears on the screen (Fig.3.8.4). The control elements of this dialog are represented in Table 3.8.7.

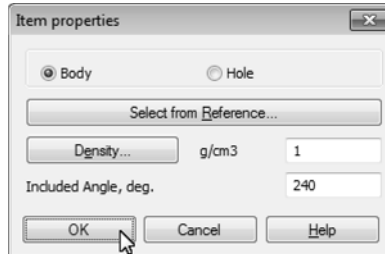


Fig. 3.8.4. Object Properties **dialog**

Tab. 3.8.7. Body/hole property controls

Element	Description
Body, Hole	Specify whether the contour is a border of a body or of a hole.
Select from Reference	This button allows selection of material from the Reference of Materials (if installed on your computer). The density value of selected material appears in the Density field.
Density	This button allows the selection of density value from density reference file*. The selected value appears in the Density field. You can also enter necessary value with the keyboard.
Inclination angle	Sector angle of a revolved body.

* File ...*ASCOMKOMPAS-3DV...*\Sys*Graphic.dns*. The user can edit material density values contained in this file.

MP calculation begins right after the first border is selected. The system defines the following values:

- ▼ mass of the body,
- ▼ volume of the body,
- ▼ mass center coordinates,
- ▼ planar moments of inertia,
- ▼ axial moments of inertia,

- ▼ centrifugal moments of inertia.

When a new border (of the hole of body) is selected, MP are recalculated.



For correct MP calculation of bodies with holes, when selecting the hole it is necessary to set up the density of material from which body with this hole is made.

3.8.3.4. MP of Extruded Bodies



To calculate MP of an extruded body with selected section, call the **MP of Extruded Bodies** command.

Extruded body for MP calculation is generated according to the following rules.

- ▼ The section of extruded body lies in the XOY plane (in the plane of the drawing).
- ▼ Extrusion is made along Z axis.

Set up borders of body as XOY plane.

Upon selection of each contour, object properties dialog appears on the screen (Fig.3.8.5).

The body width in extrusion direction is to be entered in the **Width** field. Other control elements of this dialog are shown in the Table3.8.7.

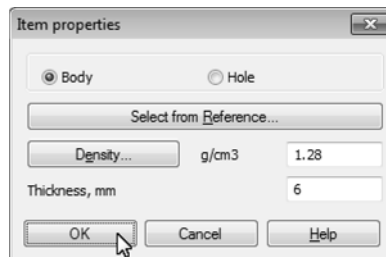


Fig. 3.8.5. Object Properties **dialog**

MP calculation begins right after the first border is selected. The system defines the following values:

- ▼ mass of the body,
- ▼ volume of the body,
- ▼ mass center coordinates,
- ▼ planar moments of inertia,
- ▼ axial moments of inertia,
- ▼ centrifugal moments of inertia.

When a new border (of the hole of body) is selected, MP are recalculated.

3.9. Utilities

3.9.1. Automatic object numbering

3.9.1.1. Objects to be numbered

In KOMPAS-3D graphical documents, the following objects are numbered automatically:

- ▼ tables,
- ▼ reports,
- ▼ tables of versions,
- ▼ reports on array with the table of modifiable variables,
- ▼ modification symbols.

In case of automatic modification numbering, the *Table* or *Modification symbols* are placed into a numbering group (the maximum number of objects in each group is 2,147,483,647). The object number is a link to its number in the group.

3.9.1.2. Numbering groups

A numbering group includes the objects with the same numbering parameters according to the group's settings. The object numbers are not always consequent (i.e. some numbers may be missing), but not duplicated.

Each object can be assigned only one numbering group.

You can use multiple automatic numbering groups (both *system* and *custom* ones) in a single document.

System groups are created in documents by default. The system group's name begins with the # symbol. It is impossible to add or delete a system group or change its name.

User groups are added and deleted by the user in the process of working with the document. You can enter or edit any group name.

You can set up object numbering parameters in each group (both system and user). Numbering group parameter setup, as well as creation/deletion of user groups can be made in the following dialogs: **System group numbering parameters** and **User group numbering parameters**.

Working with the dialogs is described in Section 9.2.6.22 on p. 1832.

3.9.1.3. Numbering principles



To work with groups, you can use the **Numbering** window available from **Maintenance — Numbering**. The window contains the drop-down list of numbering groups, the list of the current group's objects and a toolbar. For each object in the group, the list displays the type and number.

The window's elements are available, if no command is run.



By default, the list of the group's objects is displayed in the ascending order. The **Sorting** button in the toolbar of the **Numbering** window should be enabled. You can sort the list in the descending order of the object numbers, if necessary. To do this, simply disable the button.



To access the group configuration dialog, click the **Parameters of the numbering groups** on the toolbar in the **Numbering** window.

For information on how to edit them, see 3.9.1.3.1–3.9.1.3.4.

3.9.1.3.1. Adding an object into a numbering group

You can add an object in a numbering group in several ways.

- ▼ When creating or editing an object, choose the required numbering group from the **Numbering** list on the Properties panel (on the **Table** tab for tables or on the **Symbol** tab for modification symbols). This will add an object to a group you've selected.



- ▼ Once the required objects are created, choose a group you want in the **Numbering** window. In the document windows that displays, choose the objects to be added to the selected group. Click **Add selected objects** on the toolbar in the **Numbering** window.

If an associative table is split into several parts, each part is numbered separately. You can disable the numbering for some tables, if necessary (see Section 3.9.1.3.2 on p. 1278).



If you've specified a name when creating or editing a table (or typed any text in the **Text input** dialog), this name or text will be preserved when the modification symbol is added to a numbering group.



A single object cannot be added into multiple numbering groups at once. As such, if you add an object into a group and this object is already in another group, then the object will be excluded from the first group and included into the second one.

The added object is placed at the end of the group object list and is assigned the last number in the list. To change a number of an object, simply move an object up or down in the list (see Section 3.9.1.3.4 on p. 1279). If there's a free number in the document, you can assign it to the newly added object (see Section 3.9.1.3.3 on p. 1279).

3.9.1.3.2. Excluding an object from a numbering group



You can exclude an object from a numbering group in several ways.

- ▼ To do this, select an object in the list of the group's objects and click the **Exclude from group** button on the **Numbering** window's toolbar.
- ▼ When creating or editing an object, choose **Do not use** from the **Numbering** list on the Properties bar (on the **Table** tab for tables or on the **Symbol** tab for modification symbols). This will disable the numbering for the selected object.
- ▼ When editing an object, remove the link number from the modification symbol in the text input dialog (for tables, see Section 4.2.2.1.3 on p. 1364; for modification symbols, see Section 3.3.7.8.1 on p. 1036).



Aside from the link number, an object can contain any manually entered text. This text is not deleted in case of the link number deletion.

In its turn, the link number may contain a text line set during the numbering setup. This line is part of the number and will be deleted with it.

Once you exclude an object from a group, the link number of the matching object in the document will be removed, and the numbering is automatically rebuilt by default. If the automatic rebuilding is disabled, then the numbers of the remaining objects will not be changed. The free numbers are shown without link numbers in the list (see Section 3.9.1.3.3).

3.9.1.3.3. Free numbers and automatic numbering rebuilding

If an object is deleted from a group, the number assigned to it may be deleted from the list of group's objects or maintained as a *free* number. It depends on the value of the **Auto-rebuilding of numbering** parameter, which is set during the numbering setup (see Section 9.2.6.22 on p. 1832). If **Yes** is selected (auto-rebuilding is enabled), exclusion of the object from the group will result in automatic rebuilding of object numbers in the group that are positioned after the excluded object within the list. There will be no free numbers. If **No** is selected (auto-rebuilding is disabled), group's object numbers will not be modified; lines with free numbers will remain in the list.



You can create any number of lines with free numbers in the list. To do so, disable the auto-rebuilding of the numbering and move the last object in the list down for the required number of positions.



If a final object in the list is excluded, its number in the list will be deleted irrespective of the value of the **Auto-rebuilding of numbering** parameter, i.e. there will be no line with a free number.



The list's line with free numbers may be deleted manually (they are deleted in the same way as objects are deleted from groups) or automatically by enabling auto-rebuilding of the numbering.

3.9.1.3.4. Changing a number



In order to change the number of a designation, change its position in the list of the numbering group's objects using the **Move down** and **Move up** buttons. Each click of the button moves the selected object one position down/up. It will occupy the adjacent line of the list and get the number of the object contained in the line, i.e. objects swap positions and numbers with each other. The same operations are performed if there is a line with a free number next to the movable object instead of another object.



You can move both the lines with objects and the lines with free numbers. If a line with a free number is moved to the last position in the list, it will be automatically deleted.

Note the following features when moving an object.

- ▼ The first object in the list can only be moved down.
- ▼ The last object in the list can be moved down depending on the value of the **Auto-rebuilding of the numbering** parameter (see Section 9.2.6.22 on p. 1832). If **Yes** is selected, moving down is not available. If **No** is selected, the object can be moved down. The object is moved to the next position in the list and gets the respective number; the line in which the object had been previously positioned remains empty while the initial number of the object becomes a free number.

3.9.2. Named groups

During the edition of a drawing, you often need to combine, for a while, separate image elements logically connected with each other, for convenience of their search and editing. For example, if you develop a design of a construction with a cover, it will be convenient to select and replace an image of a cover and all structural elements located on it at the same time if the height of the construction changes.

When using KOMPAS-3D, you can combine an arbitrary number of objects into **groups**. The essential fact is that the inclusion of an object into a group does not impose any restrictions on its independence. The object can still be edited individually, up to its complete deletion.

Each object of a graphic document can be included in several different groups created in this document.

For operation convenience, when creating each group, be sure to set its name, which is subsequently displayed in a list for group selection while different operations are performed.

The group can include the objects located on different layers (see Section 3.5.6 on p. 1181) and in different views (see Section 3.5.4 on p. 1160) of a drawing.

The following editing commands can be applied to the group:

- ▼ **Shift** (see Section 3.4.2 on p. 1120),
- ▼ **Rotation** (see Section 3.4.4.1 on p. 1129),
- ▼ **Scaling** (see Section 3.4.4.2 on p. 1130),
- ▼ **Symmetry** (see Section 3.4.4.3 on p. 1131)
- ▼ **Copying** (see Section 3.4.3 on p. 1122).



When the commands **Symmetry** and **Copy** are called, no new groups are formed in a document. Only copies of the objects contained in the source group are created.

Groups can be also copied and transferred via a clipboard (see Section 3.1.3 on p. 876).

A group can be ungrouped at any time; that will in no way affect the member objects.

The object groups are operated in the dialog (Fig.3.9.1), called by the **Service— Groups...** command.

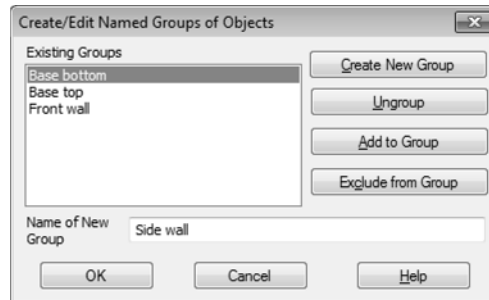


Fig.3.9.1. Group operation dialog

3.9.2.1. Creating a New Group

To create a new named group, do the following.

1. Select the objects to be included in a group (the methods of selecting objects are described in Section 3.1.3 on p. 876).
2. Call the Group Operation dialog.
3. Enter a new name for the group and click the **Create New Group** button.

3.9.2.2. Adding Objects to a Group

To include additional objects into an existing named group, perform the following operations.

1. Select the objects to be added in a group.
2. Call the Group Operation dialog.
3. In the list, select the name of the group, to which the selected objects shall be added, and click the **Add to Group** button.

3.9.2.3. Excluding Objects from a Group

To exclude some of the member objects from a named group, complete the following steps.

1. Select the objects to be excluded from a group.
2. Call the Group Operation dialog.
3. In the list, select a name of the group, to which the selected objects belong, and press the **Exclude from Group** button.

3.9.2.4. Selecting a Group

A group must be selected before calling editing commands. To select a named group in a document, perform the following operations.

1. Call the **Select — Group** command. A Group Specifying dialog will appear on the screen.
2. Select from a list the name of the group (names of groups) to be selected.

All objects forming the specified group (groups) will be selected.

3.9.2.5. Ungrouping

If the named group is no more required, you can ungroup it. Ungrouping a group has no effect on the member objects.

To ungroup, do the following.

1. Call the Group Operation dialog.
2. In a list select the name of the group (names of groups), which must be ungrouped, and press the **Ungroup** button. The name (names) of the selected group (groups) will be removed from a list.

3.9.3. Using Bitmap Images

You can insert images from external bitmap files into graphic documents; these bitmap files have the following formats:

- ▼ BMP,
- ▼ GIF,
- ▼ JPEG,
- ▼ PNG,
- ▼ TIFF,
- ▼ TGA.

The insertion of bitmap objects is described in Section 3.9.3.1.

A bitmap image inserted in a KOMPAS-document can be saved in a file. To do it, select the insertion and apply the **Save Bitmap Object As...** command from a context menu. In a dialog which appears on a screen, select the type of the file and set its name and location.

Options for editing the inserted image (See Section 3.9.3.2) and its cutting by contour (See Section 3.9.3.3) are also available.

If necessary, you can set up editing of bitmap images inserted into documents, see Section 9.1.7.13 on p. 1715.

3.9.3.1. Insertion








To insert a bitmap image, call the **Insert— Picture** command.

In the dialog that pops up, specify the desired source file of the insert.

The boundaries of an image to be inserted will appear in the current document and on the Property bar— the elements controlling insertion. These elements are presented in Table 3.9.1.

Tab. 3.9.1. Elements controlling the insertion of a bitmap image

Element	Description
Source File	Full name of the image source file.
 Change the Source	A button allowing the selection of another file as an image source.
 Method of Insertion	A group of switches making it possible to select an insertion method (see Table 4.1.13 on p. 1317).
 Resolution	A field for entering the resolution of the image to be inserted. It is available on the Property bar if the source file contains no information on the image resolution.
Base point	These are the coordinate fields of the base point.
Angle	The field for entering the angle of the image rotation in the current coordinate system.
Scale	This is the field of the image scaling factor.
Information	A panel containing data about the bitmap image to be inserted. If the values in the fields Resolution and Scale are changed, the values in the lines Width, mm and Height, mm of this panel are recalculated.
 Mode	A group of switches for selection of the mode to display a bitmap object: partially or wholly. This group shall become available after the definition of the picture cutting border (See Section 3.9.3.3 on p. 1284).
	

After definition of the base point the insertion ghost shall be fixed. With a single command call, you can insert the selected image several times, while changing its scale and rotation angle, if necessary.



You can insert a bitmap image via the clipboard. In this case, insertion controls listed in Table 3.9.1 will be available in the Property bar, except for as follows: **Source File**, **Change Source**, and **Insertion Method**.

3.9.3.2. Editing

The editing of bitmap images by means of KOMPAS-3D is impossible. To edit a picture, you need to open it in an application that can work with bitmap graphics.

To do it, select an inserted bitmap object and apply the **Edit Source** command from the context menu. This command operates in different ways depending on the method of insertion (see Table 4.1.13 on p. 1317) of the selected bitmap object.

- ▼ If the object was inserted using an external reference, when you select the **Edit Source** command, the source object opens in the application associated with the Windows file type of this bitmap object.
- ▼ If the object was introduced into a document, then after calling the **Edit Source** command it opens in the format and in the application, which were set up during the customization of the graphic editor (see Section 9.1.7.13 on p. 1715).

After the image is opened, introduce the required changes in it. Then save and close the image file. The introduced changes will be sent to a KOMPAS-document.



During editing the source of the bitmap object introduced into a document, the file, the type of which is specified during customization of a graphic editor and the name of which is the same as the name of the insertion source file, is created. The created file is located in a folder intended for saving temporary files. If a picture is closed before the completion of a KOMPAS-3D working session, the changes in the picture are sent to a KOMPAS-document and the temporary file is deleted. Otherwise, the image in the KOMPAS document does not change and the image file is saved in a folder for temporary files.

If the object was inserted by means of external reference, its context menu contains the command **Edit with...** After applying this command, a dialog for specifying a program appears. Select the program supporting the file type, which has the inserted picture, and press the **OK** button. The picture will be opened by the specified program.

You can change the location of a picture in a document and its scale, moving its defining points with a mouse. In addition, you can call the following object editing commands to the pictures:

- ▼ **Shift** (see Section 3.4.2 on p. 1120),
- ▼ **Rotation** (see Section 3.4.4.1 on p. 1129),
- ▼ **Scaling** (see Section 3.4.4.2 on p. 1130),
- ▼ **Symmetry** (see Section 3.4.4.3 on p. 1131)
- ▼ **Copying** (see Section 3.4.3 on p. 1122).



Note that the result of performing the two last commands will be the insertion of additional pictures into the document.

You can also copy figures using a mouse (see Section 3.4.1.1.2 on p. 1114) or via the clipboard (see Section 3.1.3 on p. 876).

3.9.3.3. Cut-off

When not the whole picture is required to be displayed in the drawing or fragment, but only their part, you can hide parts of the image using the **Trim** command. The picture cut is performed along the closed line — trim border. The cut's border can be created right during the command performance or it can be preliminary built in the document.



The preliminary created cut's border may include various geometric objects while during the command performance it is possible to build only a polyline.

After the definition of the cut's border for the picture, you can select a picture display mode — wholly or partially (See Section 3.9.3.3.2 on p. 1286).

The cut's border is kept inside the picture insertion. Due to this, it moves, turns and is copied together with the picture. In case of changing the insertion dimensions (using the **Scaling** command, as the result of raster resolution changing, etc.) the border will be scaled.

To cut the picture along the other border repeat the cut-off having defined or created a new contour. Since the cutting border may be only one, a new contour will replace the previous one.

The cut's border will be removed if the insertion source changes. To cut a new picture call the **Trim** command for it.



If the previous file is specified as a new insertion source (for example, saved in the other folder), the cut's border will be retained.

When selecting a partially displayed picture, the outline frame will be formed according to dimensions of the displayed picture part. The defining points of the contour — the cut's borders are not shown.

While creating other objects, it is possible to use the defining points and cut's border lines for snapping.

3.9.3.3.1. Picture cut-off

Requirements to the cut's border:

- ▼ the border must be a circle, ellipse, rectangle, polygon, closed polyline or closed contour (the contour must not contain Bezier or NURBS curves),
- ▼ the border must cross the image,
- ▼ the border having self-intersections shall be located fully within the picture borders.

If required, create a cut's border — geometrical object or contour which meets the above mentioned requirements.

Perform the following actions to cut a picture.



1. Call the **Editor — Trim** command.
2. Specify the picture for cutting double clicking the mouse on it (when the cursor passes above the pictures inserted into the document they are highlighted with the outline frame).



The picture for cutting may be defined before the command calling by its selecting. In this case the **Trim** command is convenient to be called from the context menu.

3. Set the cut border.
 - ▼ To specify the previously created border, click it with the mouse (when the cursor passes over the objects which may be used as a border they are highlighted).



The **Mode** group of switches allows you to specify if the selected object must be deleted or not after the border is created.



No link is formed between an image and the selected object, i.e. deleting or editing object does not affect a cut border.



- ▼ To create a cut's border, press the **Manual Drawing of Borders** button on the Special Control Panel.

The system will switch to the manual mode of border drawing (See Section 3.2.13.1.1 on p. 956). Make the cut's border which will meet the above mentioned requirements.



Click the **Create Object** button on the Special Control Panel to continue performing the **Trim** command.



4. If you need to change a border, click the **Reselect Contour** button on the Special Control Panel and select another object or click the **Manual Drawing of Borders** button and create another border. If required, activate a full image display mode using the **Mode** switch group on the Properties bar.
5. To confirm the picture cutting press the **Create Object** button.



After cutting one picture the system will await the following one to be specified. To finalize the command, pres the **Interrupt Command** button.

3.9.3.3.2. Selection of the picture displaying mode

Picture displaying mode — wholly or partially — may be selected using one of the following two methods.

Method 1

1. Select the picture which was cut and call the **Trim** or **Edit** command from the context menu.
2. Select the displaying mode using the **Mode** group of switches on the Property toolbar.
3. Click **Create Object** button.

Method 2



1. Enable the **Properties** window displaying if it was disabled. For this, you may call, for example, the **Editor — Properties** command.
2. Select the picture for which the cutting was performed.
3. In the **Properties** window find the **Drawing Mode** property. Click in the property value cell, open the list and select the required line from it.



This method allows changing the displaying mode of several selected pictures. For more details about the **Properties** window and changing of the object properties, see Section 3.4.1.2 on p. 1117.



If the border is outside the picture boundaries (it may occur because of changing of the picture area during its editing in the graphic editor), it will not be removed from the insertion, but the mode of the picture displaying will automatically switch to the whole one. To cut this picture again, call the **Trim** command and create or specify a new contour for it.

4. Working with Texts and Tables

4.1. Text editor

4.1.1. General Information

Text editor is a component of the KOMPAS-3D system. Its application area is mainly developing various text and graphics documentation. Documents can be laid out according to standards or be of an arbitrary format.

While creating documents, it is possible to use any fonts available in Windows, both vector and TrueType fonts.

The KOMPAS-3D delivery suite includes TrueType fonts whose typeface complies with GOST2.304-81 of the Unified System of Engineering Drawings. Drawing fonts:

- ▼ **GOST type A** (font file name — *gost_a.ttf*),
- ▼ **GOST type B** (font file name — *gost_b.ttf*),
- ▼ **GOST type AU** (Unicode font, font file name — *gost_au.ttf*),
- ▼ **GOST type BU** (Unicode font, font file name — *gost_bu.ttf*),

Besides, the distributive disk includes True Type fonts that contain special symbols:

- ▼ **Symbol type A** (font file name — *symbol_a.ttf*),
- ▼ **Symbol type B** (font file name — *symbol_b.ttf*).

Delivery set also includes vector fonts that contain the same symbols as True Type fonts:

- ▼ **GOST type A (potter)** (font file name — *gost_a.fon*),
- ▼ **GOST type B (potter)** (font file name — *gost_b.fon*),
- ▼ **Symbol type A** (font file name — *symbol_a.fon*),
- ▼ **Symbol type B** (font file name — *symbol_b.fon*).

Basic capabilities of text editor:

- ▼ Setup of font and paragraph parameters;
- ▼ Work with text styles (use of ready fonts and development of own ones);
- ▼ Auto Text capability for entering frequently used text fragments (text templates);
- ▼ Entering of special signs and symbols (form tolerances, extreme deviations, designations for welded joints, etc.) from the system library;
- ▼ Entering formulas, ratios, subscripts and superscripts;
- ▼ Creation of lists with various degrees of inclusion;
- ▼ Creation of random tables;
- ▼ Outputting ready documents to a plotter or printer;
- ▼ Spell checking of texts in different languages.

The users who have some experience with a popular text editor MS Word for Windows (or another text processor for this operating system), will find many familiar functionalities for processing text in KOMPAS-3D and will easily master the system.

Text processor is used to enter and process graphic text information in the following modes:

- ▼ Creation of technical requirements, filling in the basic inscription on drawings,

- ▼ Creation of various inscriptions and tables (including if they are integral parts of dimensions and designations) in drawings and fragments,
- ▼ Creation of separate text and graphic text documents,
- ▼ Creation of tables for basic inscriptions of drawings, BOM and text documents.

Each of these modes has some differences in terms of interface and set of available commands. In general, working methods and principles to set parameters are the same in all the modes.

In future, description of any given capability of text processor will stipulate differences (if available) for its use in various work modes.

Text processor command call can be performed using the Main menu, Tool panels and tabs of the Properties panel. For most commands all the three call methods are available.

4.1.1.1. Text Cursor and its Control

Text cursor is a vertical blinking bar. It shows where the text will be entered or a picture/table will be inserted at the next moment. The height of the cursor is the same as that of the font. If Italics mode is selected, the cursor will be slanting; for non-italic fonts it will be upright.

Text cursor is displayed in the work field of a text document right after the document was created or loaded. The paragraph where the cursor is located is considered to be current.



In the field of a drawing or a fragment text cursor appears in a special frame once the inscription (see Section 4.1.4.1 on p. 1333) or tables (4.2.2.2 on p. 1365) creation command was called.

The position of text cursor in the current document window remains unchanged while switching to other windows. During the document loading the cursor is automatically put at the beginning of the text.

Text cursor can be moved using both the mouse and the keyboard. To control the cursor, use the keys given in Table 4.1.1.

Tab. 4.1.1. Text cursor control keys

Key/Keys	Purpose
<↶>	One position to the left
<↷>	One position to the right
<↵>	One line down
<↶>	One line up
<Page Up>	To the upper line of the current window
<Page Down>	To the lower line of the current window
<Home>	To the beginning of the line

Tab. 4.1.1. Text cursor control keys

Key/Keys	Purpose
<End>	To the end of the line
<Ctrl>+<↶>	To the beginning of the previous word
<Ctrl>+<↷>	To the beginning of the next word
<Ctrl>+<↶~>	To the beginning of the current paragraph (if the cursor is already at the beginning of that paragraph, it will move to the beginning of the previous paragraph).
<Ctrl>+<↷~>	To the beginning of the next paragraph. From the starting position of the last paragraph the cursor moves to the end of this paragraph.
<Ctrl>+<Home>	To the beginning of the first paragraph.
<Ctrl>+<End>	To the end of the last paragraph.
<Tab>	Shift to the right to the preset tabulation value
<Enter>	Start the following paragraph
<Ctrl>+<Enter>	Start the following paragraph at a new page (i.e. to insert a hard page break).
<Shift>+<Enter>	Start a new line in the current paragraph (i.e. to insert a hard line break).

When the keys <Page Up> and <Page Down> are used, the cursor is set to the current position of the first or the last line of the window correspondingly. While moving to the necessary page using **Current Page** field (see Section 1.4.1.5 on p. 79), the cursor is set to the current position of the first line of the indicated page.

When the text is scrolled using the mouse, the position of the cursor in the text remains unchanged.

In the text input mode in the drawing the keys can only be used for the current text fragment (it is limited by a thin overall frame). To switch to this mode, double-click the mouse on the required inscription. The cursor remains in the position where it was at the moment of double-clicking the mouse.

4.1.2. General Working Techniques

This Section is devoted to work methods used to enter and edit most inscriptions in KOM-PAS-3D documents.

4.1.2.1. Font Selection and Setup of its Options

For preparation of design and process deliverables under USED, standard drawing fonts shall be used and actually these very fonts are used in the text styles included in the system. Though for artistic applications like posters, prospects and the like, different fonts can be used.



To connect new fonts to Windows, use the **Control Panel – Fonts** system utility.

As KOMPAS-3D is mainly designed for production of technical documentation, setup of text parameters is slightly different from that established in universal text processors.

The main difference is assignment of height for symbols, intervals and distances between lines in millimeters (as it is usually done in drawings) rather than in points.

In the mode of work with the text the current font name and the example of its pattern are displayed in the corresponding fields of the Properties panel.

To set parameters for the current font, call the **Font** command.

The dialog that allows to set parameters for the current font will be displayed (Fig.4.1.1).

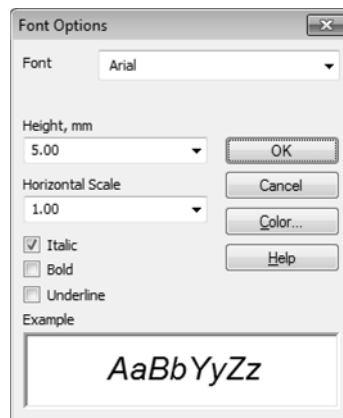


Fig. 4.1.1. Font options setup dialog

The same font parameters can be set up separately, using elements of the **Formatting** tool panel (Fig.4.1.2) or of the **Formatting** tab of Property bar (Fig.4.1.3).

The font parameters set by you will be used for text input, until you change them.

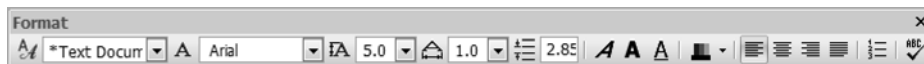
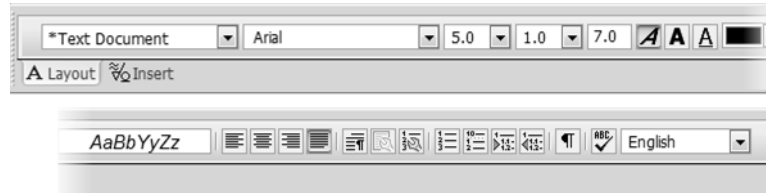


Fig. 4.1.2. **Formatting** panel

Fig. 4.1.3. **Formatting** tab

4.1.2.2. Text Editing Insertion and Replacement Modes

To edit the entered text, you should use the keys given in Table 4.1.2.

Tab. 4.1.2. Keys used to edit the text

Key	Purpose
<Backspace>	Deletion of one symbol to the left of the cursor [*] . While deleting the leftmost symbol in the line, the cursor moves to the last position of the previous line.
<Ctrl>+<Backspace>	Deletion of one word to the left of the cursor [*] .
<Delete>	Deletion of one symbol to the right of the cursor [*] . If the cursor stands before the marker of the paragraph end (see Section 4.1.2.8 on p. 1304), pressing of the <Delete> key will result in merging of two paragraphs.
<Ctrl>+<Delete>	Deletion of one word to the right of the cursor [*] .
<Insert>	Switching between the insertion and replacement modes. In the insertion mode the symbol is inserted in the cursor position, drawing apart other symbols. In the replacement mode the entered symbols replace the symbols to the right of the cursor. When the replacement mode is enabled, the displayed cursor becomes thicker to indicate this mode.

^{*} In this case, the right part of the line is moved to the left.

To insert a new paragraph into the text, put the cursor to the end of the paragraph, after which a new one is to be inserted, and press <Enter> key. If the cursor stands not at the end of the paragraph, pressing this button will result in division of the current paragraph into two separate paragraphs.

To delete a few words or lines in sequence, highlight them and press <Delete> key or call the **Editor – Delete** command.

4.1.2.3. Text Fragments Selection

To extend the action of some command to a part of the text (for instance, while setting height of symbols), this part is to be previously selected.

To select a random text fragment, put the mouse pointer at the beginning of this fragment, click the left mouse button and move the selection in the required direction, without releasing the button. Symbols of the selected text are displayed in inverted color.

To select one word, just double click it; to select a sentence, hold `<Ctrl>` and click it.



To select one or several lines of the text, put the cursor to the left of the input field border (Fig.4.1.5 on p. 1299) beside the first line to be selected. When the cursor changes its view, click the left mouse button and drag the cursor down till the required number of lines will be selected.

The text can also be selected using the keyboard. Put the cursor to the place from which you are going to start the selection, press `<Shift>` key and, without releasing it, press arrow keys to extend the selection in the required direction or click on the end selection position.

To select text by words, use the following key combinations: `<Ctrl>+<Shift>+<␣>` and `<Ctrl>+<Shift>+<▶>`.



To shift the selected limits, hold `<Shift>` and click the new desired position.

In order to highlight the whole text, call the **Editor — Select all** command or press `<Ctrl>+<A>`.

To cancel the selection, click the left mouse button on any place of the current window or press one of the cursor control buttons or `<Esc>`.

4.1.2.4. Copying and Transfer of the Text Using the Clipboard

Copying and transfer of the text through the clipboard can be performed both in the same document and between various KOMPAS-3D documents.

To copy or transfer a text fragment, highlight it and put it to the clipboard. Then put the cursor to the place where the fragment is to be inserted, and insert the text from the clipboard. For details about working with the clipboard, see Section 3.1.3.5.

The text fragment inserted from the clipboard will be displayed starting from the current position of the cursor. During insertion lines will be drawn apart, and the cursor will be put after the last symbol inserted from the clipboard.

If you need to put into the text several copies of the text from the clipboard, repeat the insertion command the required number of times. Of course, in this case, the cursor is to be put to a new place to insert the following copy.

To copy and transfer text fragments, the clipboard of Windows is used. Therefore, the clipboard allows to transfer the text between KOMPAS-3D documents as well as other Windows applications documents. Thus, for instance, to transfer the text from a MS Word document, highlight the required text fragment in this document, copy it to the clipboard, then switch to KOMPAS-3D and insert the text from the clipboard to your document.



Not all the text formatting parameters can be transferred through the clipboard. After the text was inserted from another application, in case of necessity, it can be formatted using the tools of KOMPAS-3D text processor.

4.1.2.5. Text Formatting

Text paragraphs have a number of parameters that define their appearance. Paragraph parameters include:

- ▼ Line spacing,
- ▼ Left and right indent,
- ▼ Intervals before and after the paragraph,
- ▼ Text alignment method:
 - ▼ To left border,
 - ▼ By Center,
 - ▼ To right border,
 - ▼ To all width within text borders.
- ▼ Tabulation parameters.

These parameters are typical for every paragraph and can always be changed.

Besides, certain font parameters can be assigned to the whole paragraph or to some of its fragments (see Section 4.1.2.1 on p. 1294).

The enumerated **paragraph parameters** together with **font parameters** can be united into **text style**. The style can be set up once and saved, and then used many times for quick document formatting (see Section 4.1.2.7 on p. 1301).

Perhaps, while processing documents, you have to use various font and paragraph parameters frequently enough. To avoid spending time to set them up while entering the text, it is recommended to type the whole document in a uniform style, using the basic font. Then, highlighting the required paragraphs or other text fragments, set the necessary font and paragraph parameters for them or assign to them certain text styles.

4.1.2.5.1. Paragraph Properties Changing

Competent setup of paragraph parameters allows to make the text document expressive and more easy to perceive.



To change the current paragraph parameters, call the **Paragraph Properties** command.

The dialog that allows to set parameters for the current paragraph will be displayed on the screen (Fig. 4.1.4). The control elements of this dialog are represented in Table 4.1.3.

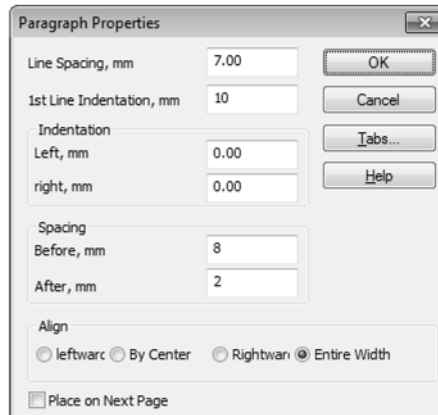


Fig. 4.1.4. Paragraph Settings Dialog

Tab. 4.1.3. Paragraph Settings Dialog

Element	Description
Line Spacing	Distance between lines in paragraph. Measurement of line spacing allows to keep proportions of distances between lines typed using a font with various symbol heights (for instance, to introduce into the text some footnotes, comments or remarks typed in a smaller font compared to the main text).
1st Line Indent	First line indent of the paragraph. This distance is measured from the text border (see Fig.4.1.5). When creating a new paragraph, the cursor is automatically set to a position with preset indent.
Indentations	Distances between the left and right border of the input field and the corresponding text borders (see Fig. 4.1.5).
Spacing	The distance between two contiguous paragraphs. It is formed by two values — the interval after the first paragraph and the interval before the second paragraph. Setting of intervals allows to separate paragraphs in a text document for their optimal perception while reading, as well as to draw attention to especially important information.
Alignment	A group of switches that allows to set the paragraph alignment method (see Table 4.1.4).
Place Next Page	An option activation of which allows to forcedly allocate a paragraph at a new page.
Tabs...	A button to call the dialog to set up tabulation parameters. Its control elements are described in Table4.1.5.

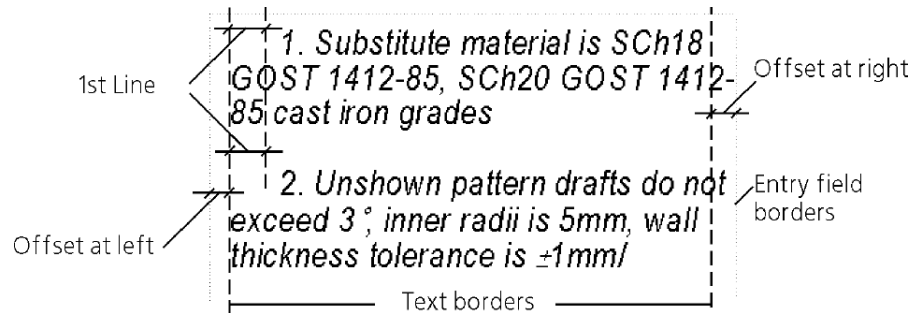


Fig. 4.1.5. Position of text at page

Tab. 4.1.4. Indentation at right

Method	Description
Left	All the lines of the paragraph start strictly at the left border of the text [*] . In this case, line ends are not aligned.
Right	All the paragraph lines end strictly at the right border of the text. In this case, line beginnings are not aligned. Typical example of text alignment to the right — address and date in letters.
Centered	The paragraph text is located symmetrically relative to text borders. This alignment method is usually applied to headers.
By all Width	Paragraph lines start at the left border of the text [*] , and end at the right border. Line expansion is produced at the expense of automatic increase in width of spaces. This alignment method is most frequently applied to the main text of the document.

* Position of the beginning of the first line in the paragraph is set by the first line indent value (see Fig.4.1.5).

Tab. 4.1.5. Control elements of Tabbing Customization dialog

Element	Description
Ref.No.	Sets the tab indents in millimeters.
Leader	A list permitting one to select a method of entering tabbing options.
Alignment	A group of switches that manages the method to align text relative to the position of tabulation (Table 4.1.6). Text alignment by tabbing can be used, for example, for constructing a table of contents.
Set	A button allowing addition of the adjustable tab in a list.

Tab. 4.1.5. Control elements of Tabbing Customization dialog

Element	Description
Delete	A button allowing deletion of the selected tab.
Delete all	A button making it possible to clear at once the entire list of tabs.

Tab. 4.1.6. Methods to align text relative to the position of tabulation

Method	Description
Left	A part of a line entered after a tab is located to the right from the tab position.
Centered	A line part entered after a tab is positioned symmetrically about the tab position.
Right	Part of the line entered after tabulation, is located to the left of its position.
Decimal point	A point that is entered in any place of a line after a tab is positioned at the tab position.

Upon setting up of paragraph parameters, close the dialog. The current paragraph and paragraphs entered after it will be processed and aligned in accordance with the performed setup. If some paragraphs of the text were highlighted before the command call, settings of formatting will only be valid for these paragraphs.

Some paragraph parameters can be set up separately, using elements of the **Formatting** tool panel or **Formatting** tabs of Properties panel (Fig. 4.1.2, 4.1.3 on p. 1295).



To move the text that follows the cursor to the beginning of a new page (i.e. to insert a page break), press `<Ctrl> + <Enter>`.

4.1.2.5.2. Symbol Case Changing

If you have forgotten to switch the case in time, i.e., you have entered a few upper-case letters instead of lower-case ones, or vice versa, select the improperly typed fragment and call the



Change to upper case command — press `<Ctrl> + <Shift> + <U>`,



Change to lower case command — press `<Ctrl> + <Shift> + <D>`.



By default, the **Change to upper case** or **Change to lower case** commands are not available in ether menu or bar. If necessary, you can add these commands to a menu or to any bar (see Section 9.4.2.1 on p. 1872).

4.1.2.6. Text Language

A document or an object may contain texts in different languages. The language is one of the text properties. The language can be attributed to the text when entering or when editing. To write specific terms like abbreviations in Latin letters, the language property is not important. What is important is to check that the font actually includes these letters. The language property is important for spell checking because here you will need to use dictionaries of a given language.

For more details about spell checking in different languages, see Section 4.1.3.1 on p. 1322.

4.1.2.6.1. Select a language

By default, the language of the text entered from the keyboard is the same as the OS Windows language (the keyboard layout language).

To enter the text in a different language, switch the keyboard layout and start typing.

The selected language will be shown in the **Language field** in the **Format** tab of the Properties panel (Fig.4.1.3 on p. 1295). If a document includes a text in a different language, e.g. if it was copied from elsewhere, it will get **No spell checking** attribute. To attribute a language to that text, select it and select a required language from the **Language** list.



The **Language** dropdown list is designed only to select a dictionary for spell checking. You cannot change the text language using this list.

4.1.2.6.2. Switching to Latin or Cyrillic Symbols

When entering a combined text that contains words both in Russian and in other languages (typed in Latin letters), it is easy to make a mistake, i.e., to type a few words in wrong symbols. The KOMPAS-3D text processor provides commands to transform symbols from Cyrillic to Latin alphabet and vice versa.



For such transformation, select the required text fragment and call the

Change to Russian register — press <Ctrl> + <Shift> + <R>,

Change to Latin register — press <Ctrl> + <Shift> + <L>.



The transformation will be performed in accordance with the keyboard layout of symbols supported by the keyboard driver installed on your computer. The text converted into Russian symbols will be automatically treated as a Russian text. The text converted into Latin symbols will be treated as an English text.



By default, the **Change to Cyrillic case** or **Change to Latin case** commands are not available in either menu or bar. If necessary, you can add these commands to a menu or to any bar (see Section 9.4.2.1 on p. 1872).

4.1.2.7. Text Styles

As it has already been mentioned before, text style is a system of paragraph and font parameters. Text style fully determines its appearance.

Of course, it is possible to format every paragraph of the text separately, changing various parameters. But it is hardly advisable during creation of large multi-page text documents with the requirements to processing that are known beforehand. For such documents, it is most convenient to use a ready style set that includes styles of common text of the document, a table cell, a table, a section header, etc.

For drawings specific text styles can be used (such as text in drawing, text in dimension inscriptions, on extension lines, text in specifications, etc.).

The KOMPAS-3D distributive disk includes ready (system) styles to create a text document as well as to enter text and special designations in the drawing. Besides, you can create your own styles using system styles as a prototype, if necessary.

Text styles can be stored:

- ▼ Directly in text documents (embedded styles),
- ▼ In style libraries (library styles),
- ▼ In sets of styles.

Each library of styles is stored in a separate file with *lts* extension. The use of styles from libraries is possible, if files of these libraries can be accessed (including through the local network) during operation.



Library styles used to create a text document, are not stored in this document. Therefore, to transfer a document to another working place or to another enterprise, it is necessary to transfer the required libraries of styles together with it.

Sets of styles are stored in the **.cfg* configuration file. After the use of a style out of the set, it is copied to the document, i.e., it becomes an embedded style.

The use of styles to create and process text documents and drawings has a number of evident advantages compared to separate formatting, as it provides:

- ▼ Uniformity to process all the documentation in strict conformity with the requirements of USED or other requirements (enterprise standards, industry standards, etc.).
- ▼ Reduction of time spent for processing of documentation.
- ▼ Reduction of time spent for correction of any element of documentation processing. For instance, to change the font in all headers, it is enough to edit the style used to create these headers, instead of formatting every header separately.

It is possible to set the so-called extended text style, in which text parameters are additionally determined (font height, horizontal spacing coefficient and line spacing) for the first three lines in the table cell.

Extended text style is mainly used to describe various cases of text input in columns of main inscriptions of the document (in cells of a table with fixed overall dimensions). In this case, you can specify text parameters for input of one, two or three lines in the column. To fill in the column with more than three lines, settings of common (not extended) text style are used.

4.1.2.7.1. Selection of Current Text Style

Name of the current text style is displayed in **Text Style** field (Fig.4.1.6).

If a few paragraphs processed using various styles are highlighted, the field is empty.

If paragraph or font parameters for the highlighted text (or any part of it) differ from those set in the style, the style name in field **Text Style** is preceded with an asterisk. To eliminate dis-conformities, i.e., to adjust text parameters to style parameters, press the <Ctrl> + <Space> key combination or apply the former style to a paragraph (paragraphs) again.

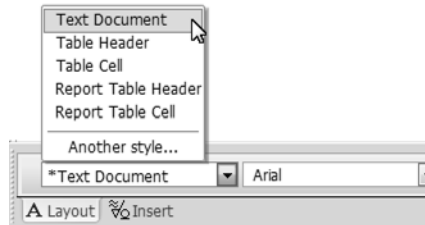


Fig. 4.1.6. Field **Text Style** in the Property bar

To select a style for the current paragraph or highlighted ones, do the following.

1. Open the list of styles, pressing the arrow button in the right part of the **Text Style** field. The list displays the names of system styles of the text, available in the active document.
2. In the list, select the desired item.

The current paragraph or highlighted ones will be reformatted in accordance with the selected style.



If the text requires use of style from the external library rather than system style, or creation and use of a new style, select the **Another Style...** line in the list of styles or call the **Text Style** command.

The dialog to select the text style will be displayed on the screen (Fig.4.1.7). The dialog tabs display lists of styles. Styles can be sorted in the lists by names or by IDs. To alter list sorting sequence click the desired column header with the mouse.

Control elements for the text style selection dialog are given in Table4.1.7.

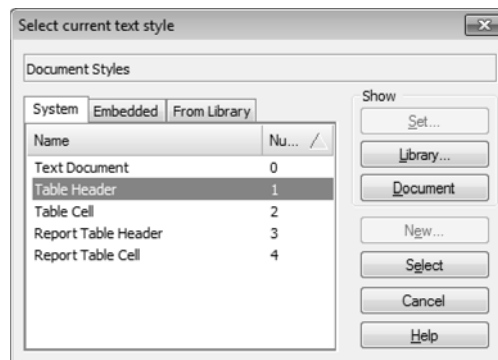


Fig. 4.1.7. Text Style Selection Dialog

Tab. 4.1.7. Text Style Selection Dialog

Element	Description
System	The field in this tab shows the list of system styles. They are available in any document KOMPAS-3D.
Embedded	The field of this tab displays the list of styles stored inside the current document. They can be created directly in the document or taken from a set.
Library	The field of this tab displays the list of text styles taken from external libraries of styles.
Show	The group of buttons used to view lists of styles (see Table 4.1.8).
New	This button is available, if tab In Memory is active, i.e., while viewing styles available in the current document or set. It allows to create a new text style that will be stored inside the current document or in a set.
Select	Allows to assign to a paragraph the style highlighted in the list. To select a style, you can also double-click the mouse on its name. Select Style dialog will close.

Tab. 4.1.8. Buttons to view groups of styles

The	Description
Set	Click this button to view styles stored in the previously created set. The content of the selected set is displayed on In Memory tab.
Libraries	Click this button to open the external library and view styles stored in it. The content of the selected library is displayed on From Libraries tab.
Document	Click this button to view system styles. They are displayed on System tab.

4.1.2.8. Formatting Symbols

Special formatting symbols are the symbols of tabulation, space, and marker of paragraph end. They are not printed out and are used for convenience of text management.

Tabulation symbol is introduced into the text by pressing the `<Tab>` key.

The space symbol is a point in the middle of line height that appears when pressing the space key.

The marker of a paragraph's end is a special symbol somewhat similar to the image of a note sign. It is introduced into the text by pressing `<Enter>` key and indicates the end of the entered paragraph. Further introduction of text will be performed in a new paragraph.



Management of described service symbols displaying is performed by the **Formatting Symbols** command.

4.1.2.9. Search and Replacement of Text

KOMPAS-3D text processor provides the opportunity of automated search and replacement of text fragments. It is possible to search and replace separate symbols, words or a random part of a line (sub-line).

4.1.2.9.1. Text search



To find a sub-line, call command **Find...**

The dialog will be displayed on the screen (Fig. 4.1.8), in which the given text is to be entered, and search criteria are to be set.

The control elements of this dialog are represented in Table 4.1.9.



Fig. 4.1.8. Dialog for setting search parameters

Tab. 4.1.9. The dialog to set text search parameters

Element	Description
Find:	The field to enter a sub-line that needs to be found in the text.
Match Case	The option that allows to take into account symbol case. If it is switched on, the search will distinguish lower-case and upper-case symbols. Thus, for instance, sub-lines «GOST» and «Gost» will be perceived as different.
Whole Word Only	Turn on this option to search a set of symbols that constitutes a separate word (not forming part of another one). This will help to avoid unnecessary reduplications in case when the entered word can be part of other words.
Direction	The group of switches that manages the search direction. If the Forward switch is active, the search is performed onward the document, and if the Back switch is active, the search is performed in backward direction.

Tab. 4.1.9. The dialog to set text search parameters

Element	Description
Search Start	The group of switches allowing to set the beginning of the search. If the From Cursor switch is active, the search will start from current position of a cursor. If the Text Boundary switch is active, the search will start from one of text borders, depending on the selected direction (from the upper border, if the Forward switch is active, and from the lower one, if the Back switch is active).
Replace...	The button to call the search dialog and replace the text (Fig.4.1.9)

To start search, click the **OK** button. The found sub-line is highlighted, and search is stopped. To continue search with the same parameters, call the **Editor— Continue** command. If the given sub-line is not found, the respective message will be displayed on the screen.



Inside the table the text search can be carried out within the current cell.

4.1.2.9.2. Text replacement

Automated text replacement allows to quickly correct a word or expression mentioned in the document several times (for instance, part designation).



To start correction, call the **Replace...** command.

The screen will display the dialog that is to be filled in with sub-lines for search and replacement (Fig.4.1.9).

This dialog differs from the one indicated in Fig. 4.1.8 only by presence of the **Replace with:** field and the **Replacement Request** option (Table 4.1.10).

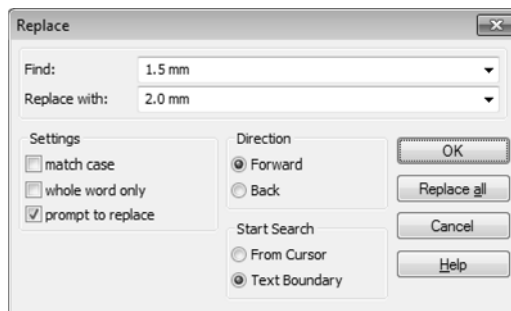





Fig. 4.1.9. Dialog for setting replacement parameters

Tab. 4.1.10. Control elements for text replacement

Element	Description
Replace with:	The field to enter a sub-line that is to be introduced into the text instead of the found one.
Prompt to Replace	Turn on this option, so that the system requests confirmation to replace every found sub-line.
<p>To start search and replacement, click the OK button.</p> <p>If all the entrances of the indicated sub-line are to be replaced at once, click the Replace All button. In this case, after replacement of each found sub-line the search and replacement will be automatically continued.</p>	
	The command to replace the text can be used to quickly delete the present sub-line from the text. To do this, enter the given sub-line in the Find: field, and leave the Replace with: field empty.
	To continue search and replacement with the same parameters, call the Editor – Continue command. Sub-lines entered for search and replacement are stored in memory until the work session is over. Therefore, in case of necessity, they can be selected from the list rather than entered again.
	Inside the table the text can be changed within the current cell.

4.1.2.10. Using text blocks

KOMPAS-3D text processor provides the opportunity to save and load text in blocks.

To save a continuous text block to a separate file on the disk, do the following.

1. Highlight the required text fragment.
2. Call the **Save Block** command.
3. In the dialog displayed on the screen set the folder and the file name for saving.
By default, the block is saved in a text document KOMPAS-3D, i.e. in the file with the *kdw* extension. In case of necessity, you can select *txt* format for recording.
To load the text from a **.kdw*, **.txt* or **.rtf* file to the document, do the following.

1. Put the cursor to the place where the text block is to be inserted.
2. Call the **Load Block** command.
3. In the dialog that appears, specify the folder and the file name. The text contained in the selected file will be inserted into the active document.

4.1.2.11. Special Inserts

As KOMPAS-3D text processor is specially designed for development of technical documentation, it contains commands that allow to insert various objects into graphic text documents and inscriptions in drawings.

Commands that allow to realize special inserts are grouped in the **Insert** menu, and buttons to call them — in the **Insert to text** bar (Fig. 4.1.10).



Fig. 4.1.10. **Insert to text** panel

Besides, commands can be called using control elements located in the **Insert** tab of Property bar (Fig. 4.1.11).

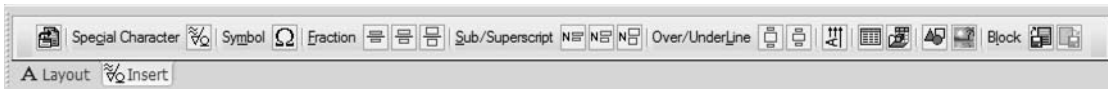


Fig. 4.1.11. **Insert** tab

4.1.2.11.1. Fractions

You can introduce into the text fractions of various height — full, average, and small. They differ from each other by height of symbols in numerator and denominator (Table 4.1.11).

Tab. 4.1.11. Size of symbols in fractions and indices of various height

Object height	Height of symbols from height of symbols of the current font, %
Full	100
Middle	≈67
Small	≈45

Upon calling the fraction insertion command the fraction line appears in the current position of the cursor, while the cursor itself moves to the position to enter numerator. Shift to enter denominator is performed by clicking the mouse under the fraction line or by pressing the <▶> key, when the text cursor stands at the last position in numerator.

While entering a fraction, numerator and denominator are centered automatically. Size of the fraction line always corresponds to the expression of the maximum length (in numerator or denominator).

While moving the cursor with the use of keys <▶> and <↶> along the line that contains a fraction, the following rules are valid.

- ▼ During left-to-right motion the cursor goes first through numerator, and then through denominator.
- ▼ During right-to-left motion the cursor goes first through denominator, and then through numerator.

It is possible to insert fractions into numerator and denominator of the existing fraction by creating inserted ("multi-story") fractions.

If before calling the fraction insertion command a fragment of the line was highlighted, it is transformed into the fraction numerator. Denominator is to be introduced by the common way.

$$\text{Circle} \frac{5-4 \text{ GOST } 7417-75}{40\text{HNMA N-M-B GOST } 4543-71}$$

Fig. 4.1.12. Example of using a fraction of full height

4.1.2.11.2. Indices

You can introduce into the text superscript and subscript indices of various height — full, average, and small. They differ from each other by height of symbols (Table 4.1.11 on p. 1308).

Upon calling the index insertion command, the cursor moves to the position to enter the superscript. Shift to enter the subscript is performed by clicking the mouse at the position of subscript or by pressing the <►> key, when the text cursor stands at the last position in the superscript.

While moving the cursor with the use of keys <►> and <◄> along the line that contains a fraction, the following rules are valid.

- ▼ During left-to-right motion the cursor goes first through the superscript, and then through the subscript.
- ▼ During right-to-left motion the cursor goes first through the subscript, and then through the superscript.

While inserting an index, the place is reserved for both indices, i.e., superscript and subscript. In practice it is often required to enter only one index. In this case, the position for another index should just be left empty. In future, you can fill it in simply putting the cursor to the position of the required index rather than calling the insertion command again.

Indices are automatically aligned to the left border, which is considered to be the position of the cursor before the beginning of insertion of indices.

Insertion of indices in positions of existing indices, i.e., the creation of inserted indices, is possible.

If before calling the index insertion command a fragment of the line was highlighted, it is transformed into the superscript. The subscript can be introduced by the common way.

$$a_k^m^2$$

Fig. 4.1.13. Example of the use of indices

4.1.2.11.3. Superscripts and Subscripts

Insertion of superscripts and subscripts is a kind of insertion of indices. The difference is that the base line can have various height, and superscripts and subscripts are centered relative to the base line.

You can enter a subscript with the normal base, whose symbols height is equal to symbols height of the current font, and a subscript with the increased base, whose symbols height is $\approx 150\%$ from symbols height of the current font. Height of indices is constant and is $\approx 45\%$ from symbols height of the current font.

The application area of insertion commands for superscripts and subscripts is writing various designations, mathematical formulas, etc.

Upon calling of the superscript/subscript insertion command the cursor remains where it is, and now the base line can be entered. Shift from input of the base to input of the superscript is performed using the $\langle \Rightarrow \rangle$ key, when the text cursor stands at the last position of the base, and shift from the superscript to the subscript is also performed using the $\langle \Rightarrow \rangle$ key, when the text cursor stands at the last position of the superscript.

While moving the cursor with the use of keys $\langle \blacktriangleright \rangle$ and $\langle \blacktriangleleft \rangle$ along the line that contains super/subscripts, the following rules are valid.

- ▼ During left-to-right motion the cursor goes first through the base line, then the superscript, and finally the subscript.
- ▼ During right-to-left motion the cursor goes first through the subscript, then the superscript, and finally the base line.

If before calling of the superscript/subscript insertion command a fragment of the line was highlighted, it is transformed into the base line. Superscripts and subscripts can be entered by the common way.

$$\sum_{i=1}^{i=n}$$

Fig. 4.1.14. Example of expression using superscript and subscript insertion

Insertion of fractions, indices, superscripts and subscripts can be used to form various expressions both separately and in various combinations.

$$\frac{m}{l} \int_0^l \frac{\partial^2 \omega}{\partial t^2} \partial x = 0$$

Fig. 4.1.15. Example of formula introduced by several insertion commands

4.1.2.11.4. Special Characters and Designations



To insert a special character (for example, technological design designation) into the text, call the **Special Character** command.

The special character selection dialog will be displayed on the screen (Fig.4.1.16). Special signs are grouped by sections. To view the content of the section, expand it. The sign or designation, the name of which is highlighted in the list, is displayed in the view window.

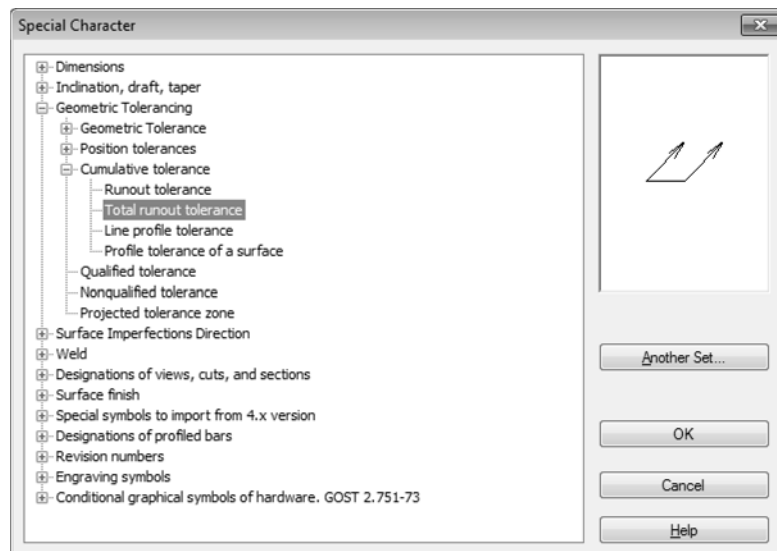


Fig. 4.1.16. The dialog to select a special character

To insert a highlighted special character, click the **OK** button. You can also double-click on its name with the mouse. The dialog will be closed, and the selected character inserted into the current position of the cursor.

By default, characters and designations stored in *graphic.sss* file (a text file that contains descriptions of special characters) are used. It is supplied together with the system and located in sub-catalog `\Sys` of the main KOMPAS-3D catalog.

To connect another *.sss file, click the **Another Set** button in the special sign selection dialog.

4.1.2.11.5. Symbols



To insert a special symbol into the text, call the **Symbol** command.

The dialog that contains the table of symbols will be displayed on the screen (Fig.4.1.17).



Fig. 4.1.17. Example of symbols table

Select a font and a set of symbols from the corresponding lists. To accelerate choice, you can press the key with the first letter of font name or of set of symbols.

The cell of the current symbol in the table is displayed with the black background. You can move along the table of symbols both using keys with arrows and the mouse.

To insert the selected symbol in the current position of the cursor, click the **OK** button. You can also double-click the left mouse button on the required cell of the table.

The dialog will be closed, and the selected symbol will be inserted into the text.



KOMPAS-3D consists of six TrueType fonts and four vector fonts, specially designed to enter texts in accordance with USED. Names GOST type A, GOST type AU, GOST type B, GOST type BU, Symbol type A, and Symbol type B are assigned to fonts. Tables of these fonts contain various design and technological signs, Latin figures, letters of Greek and other alphabets.

4.1.2.11.6. Text Templates

Text templates are text fragments stored in **.tdp* external files. They contain various typical texts and are used to automatize input of frequently used lines or designations.



To insert a template into the document, call the **Text Templates** command.

The Text Templates Librarian window will appear on the screen (Fig. 4.1.18).

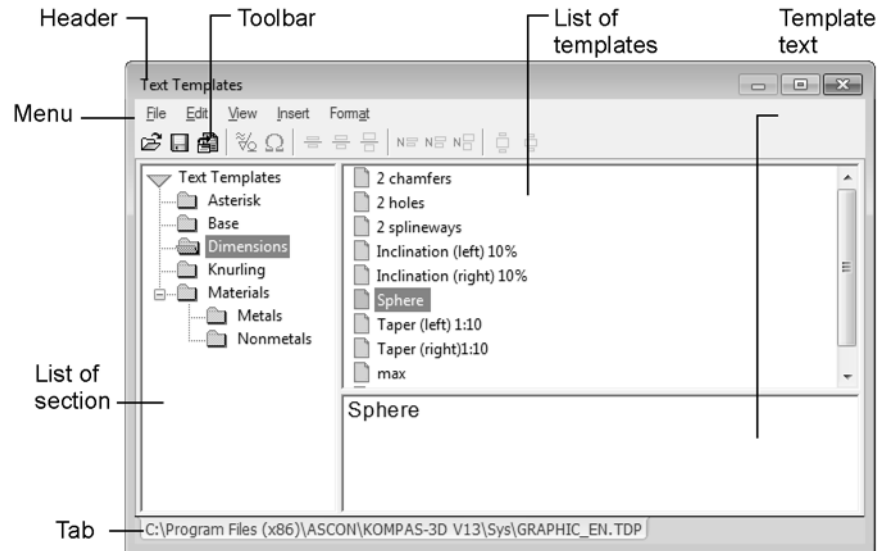


Fig. 4.1.18. Text Templates Librarian Window

Each Librarian page corresponds to one template file (*.tdp).

Text templates are grouped by sections, the list of which is displayed in the left part of the insert. The right part of it displays the list of templates for the highlighted section (above) and the full text of the highlighted template (below).

You can find the right template by viewing sections. You can use search as well. To do this, call the **Edit— Search...** command from the Librarian menu. In the dialog displayed on the screen, enter the given text fragment, set up search criteria and press the **Search** button.

To insert the right template into the document, mark it, i.e., click on the icon of this template in the list. The template will be marked with a red checkmark.



Then click the **Insert into Document** button on the ?Text Templates Librarian Toolbar. In case of necessity, you can mark and insert several templates (including those that belong to different sections and subsections).

Another way to insert a template is to double-click on its name with the mouse. In this case, the required template is not to be marked beforehand; however, you can insert only one template.

The inserted template can be edited as a common text (see Section 4.1.2.2 on p. 1295). Information about the file the template was inserted from is not saved. Due to it, you don't have to bother about transferring files of text templates to move documents to other working places.

By default, the window of Librarian for text templates opens *graphic.tdp* file. This file is supplied together with KOMPAS-3D and located in sub-catalogue \Sys of the main system catalog. You can edit the given file of text templates or create your own one. To create a new *.tdp file, do the following.

1. Call the **File — Create** command from the text template library.
2. Form the file structure (sections and subsections) using the **Insert — Section** command.
3. Create templates in sections using the **Insert — Template** command.
4. Enter template texts. In this case, it is possible to copy texts through the Windows clipboard. For special inserts, you can use buttons in the Librarian Properties panel.
5. Having finished creation of a file of text templates, save it using the **File — Save** command. In the future, to get access to a user file of text templates, it is necessary to call the **File— Open** command of Librarian.

4.1.2.11.7. Vertical Text



To insert into the document a text, lines of which are located vertically, call the **Vertical Text** command.

A thin frame that limits the fragment of vertical text will be displayed on the screen.

For convenience of work, the text is entered horizontally.

Automatic line transfer is not performed, therefore, to transfer the text to a new line, it is necessary to press <Enter> key.

Other input methods, as well as methods of editing and formatting in creation of vertical text, are the same as during work with a common horizontal text.



Once the required text is typed, exit the mode to enter vertical text by clicking the **Create Object** button or <Ctrl>+<Enter> key combination.

The entered text is automatically vertically located. It can't be surrounded by horizontal text.

Vertical text allocation is most often required to fill in some columns in main inscriptions of documents.

To edit a vertical text, double-click it with the mouse.

The text will be displayed as horizontal in a contour frame and will become available for editing.



When inputting specifications, text on a drawing and filling in the title block, vertical text cannot be inserted.

4.1.2.11.8. Illustrations

An illustration, i.e., a KOMPAS-drawing (*.cdw), a KOMPAS-fragment (*.frw) or an image stored in a bitmap file (*.bmp, *.gif, *.jpg, *.png, *.tif, *.tga) can be inserted into the text.



A document that contains the inserted image is called **insertion source**.

- ▼ To insert a drawing or a fragment, call the **Fragment** command.



- ▼ To insert an image from a bitmap file, call the **Picture** command.

In the file opening dialog displayed on the screen, indicate the file to be inserted.

After the file is selected, the **Parameters** button will become available in the dialog. Once it is clicked, the screen will display the dialog to set up parameters to insert an illustration into the document (see below).

If necessary, set up insertion parameters, and then click the **Open** button to perform insertion.

The illustration is inserted into a separate paragraph.



You can insert a graphical object from a KOMPAS document or a bitmap image into text via the clipboard.

The inserted illustration can be moved or copied through the clipboard as a separate symbol, as well as be deleted. Wrapping text around a picture is not supported.



You can change alignment of the illustration relative to the borders of the input field, the same as for a paragraph (see Section 4.1.2.5.1 on p. 1297).



When entering specifications or text in a drawing and when filling in the main inscription, it is impossible to insert illustrations.

When entering tables in a drawing, it is possible to insert fragments and bitmap images.

Setup of insertion options

Setup of parameters to insert a graphic document or a bitmap image into the text is performed in **Insertion Parameters** dialog (Fig.4.1.19).

The control elements of this dialog are represented in Table 4.1.12.

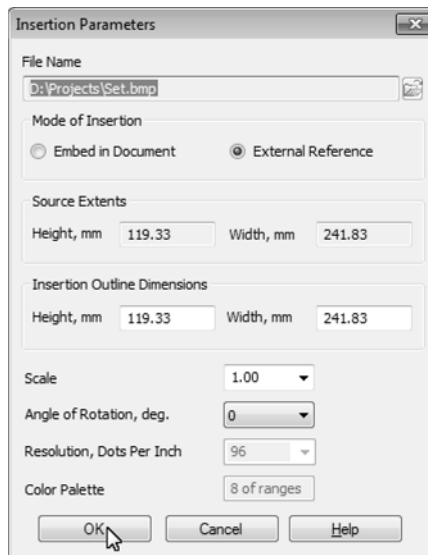


Fig. 4.1.19. Text Illustration Insert Options Dialog

Tab. 4.1.12. Dialog of parameters to paste a graphic document into text

Element	Description
File name*	This field displays the name of the file selected for insertion. A button to select another file is located to the right of the field. It is available in the course of editing and only for pictures inserted by an external link.
Method of Insertion	A group of switches that allows to select a method to insert the file (see Table 4.1.13).
Source Extents*	These fields display the height and the width of a fragment selected for insertion.
Insertion Outline Dimensions	These fields display the current height and width of the image, taking into account the scale set in the Scale field. For bitmap images, the resolution is displayed according to the setting in the Resolution field. You can enter the required value of height (width) of the insertion from the keyboard. The scale corresponding to the preset value will be defined automatically, and width (height) of the insertion will be multiplied by the received value**.
Scale	A field that allows to enter or select from the list the scale of insert. The list also contains options by width and by height that allow to fit the insert dimensions to the cell width and height. Outline dimensions of the insert are recalculated after the scale is changed.
Angle of Rotation	A list that allows to select rotation angle of the insert. Angles divisible by 90° are available.
Resolution	Only for bitmapped images. A field that displays the current image resolution. The resolution is available for change, if the source file of the insert doesn't contain resolution information. Outline dimensions of the insert are recalculated after the resolution is changed.
Color Palette*	Only for bitmapped images. A field that displays color resolution of a graphic file.

* The field is for information only and cannot be edited.

** The calculation's precision is up to 0.01. The resulting scale value is rounded downward. As a result, the final height (width) value can be different from the one initially entered.

Tab. 4.1.13. Image insertion methods

Method	Description
Embed in Document	The file content is copied to the document. The link with the source is not saved. The changes made in the source file are not reflected in the document containing the insertion. When moving this document to another workplace, the move of the insertion source is not required.
External Reference	The link to the source file of the insert is formed in the document. The changes made in the source file are reflected in all documents containing insertions. When moving these documents to another workplace, the move of the insertion source is also required.

Editing of inserted illustrations

You can change parameters to insert an illustration. To do this, double-click on the illustration inserted into the text or call the **Insertion Options...** command from its context menu. The screen will display **Insertion Options** dialog (see above).

You can edit any parameters of the insert, including change of the source file.

Note that the insertion method can be changed only for illustrations inserted by an external link — they can be included in the document. Reverse transformation is not available.

The context menu of the insert of a bitmapped image includes the **Edit Using...** and **Edit Source** commands. Their use is described in detail in Section 3.9.3.2 on p. 1283.

4.1.2.11.9. Insert report into text



You can insert model or graphical document report into text.

To do this, call the **Insert Report** command.

After calling the command, the File Open dialog will appear. Select a model document or graphic document which objects will be included into the report.

Once the selected document is opened the report creation process is started.

Perform actions provided in the **Create Report** command. More on creating reports is detailed in Section 5.2.3 on p. 1417.

After placing report in the document from which the **Insert Report** command was called, report creation and insertion is finished automatically.

The report is inserted into a separate paragraph. The inserted report table can be edited, moved or copied through the clipboard, as well as be deleted.

4.1.2.12. Lists

While developing various technical documentation (specifications, parameter lists, etc.), it is often necessary to number paragraphs, i.e., to create lists (Fig. 4.1.20). KOMPAS-3D text processor contains various tools to create diverse lists and specifications as well as to manage them.

<i>Power block</i>	<i>1. Power block</i>	<i>1. Power block</i>	<i>1. Power block</i>
<i>1. Electromotor</i>	<i>a. Electromotor</i>	<i>1.1. Electromotor</i>	<i>1.I. Electromotor</i>
<i>2. Compressor</i>	<i>b. Compressor</i>	<i>1.2. Compressor</i>	<i>1.II. Compressor</i>
<i>3. Gear box</i>	<i>c. Gear box</i>	<i>1.3. Gear box</i>	<i>1.III. Gear box</i>
<i>4. Coupling</i>	<i>d. Coupling</i>	<i>1.4. Coupling</i>	<i>1.IV. Coupling</i>
<i>5. Control console</i>	<i>e. Control console</i>	<i>1.5. Control console</i>	<i>1.V. Control console</i>
<i>6. Load frame</i>	<i>f. Load frame</i>	<i>1.6. Load frame</i>	<i>1.VI. Load frame</i>
<i>7. Shock absorber</i>	<i>g. Shock absorber</i>	<i>1.7. Shock absorber</i>	<i>1.VII. Shock absorber</i>
a)	b)	c)	d)

Fig. 4.1.20. Lists: a) without nesting, b) nested, c), d) nested with various types of numbers

You can number and align paragraphs, as shown in the pictures. However, the use of automatic numbering is much more effective both while creating and editing lists.

4.1.2.12.1. Creation and Management of Lists

There exist two methods to form lists:

- ▼ Numbering of existing paragraphs
- ▼ Numbering of paragraphs at input.

The first method is more obvious, therefore, we will first consider how to form a list out of a text that already exists. Numbering changes neither paragraph borders, nor the alignment method, nor other parameters. Therefore, to improve perception of the list, the document can be previously formatted (see Sections 4.1.2.1 on p. 1294, 4.1.2.5.1 on p. 1297, 4.1.2.7 on p. 1301).

To transform a few paragraphs into a list, do the following.

1. Highlight the paragraphs to be numbered.
2. Call the **Set Numbering** command. The serial number will appear at the beginning of the paragraph, i.e., each paragraph will turn into the list item.

The list received this way doesn't include nesting, i.e., all its items are located at the same (first) level (Fig. 4.1.20, a).

If necessary, you can move some paragraphs to the next level, i.e., increase nesting of several items of the list by one (Fig. 4.1.20, b). To do this, highlight the required paragraphs and call the **Increase Nesting** command. Numbering of the list will change correspondingly.

To perform the reverse operation, i.e., to decrease the existing level of paragraph numbering, highlight them and call the **Decrease nesting** command.

Forming of the list using the second method is performed in the following order.

1. Enter the paragraph that makes the first item of the list.
2. Call the **Set Numbering** command. Figure 1 will appear at the beginning of the paragraph.
3. Set the cursor to the last position of the numbered paragraph and press <Enter> key. A new paragraph will be created with the following number — 2.
4. To increase or decrease the numbering level of the current paragraph, call the **Increase Nesting** or **Decrease Nesting** command, respectively.

5. Enter the required number of list items, locating them at the required levels.

To cancel numbering of the current paragraph (for instance, to complete input of the list) or highlighted paragraphs (for instance, to transform the list to the common text), recall the **Set Numbering** command.



Set Numbering button in **Formatting** panel and a switch with the same name in **Formatting** tab of the Property bar indicate that the paragraph has the numbering property.

Often it is necessary to create a few independent lists in one document (or an inscription in a drawing). To start numbering from the beginning, i.e., from the first level and from the first number, do the following.



1. Put the cursor into the paragraph that makes the first item of the new list.
2. Call the **New List** command.

This command covers only the current paragraph, therefore, it is available if there are no highlighted paragraphs.



In principle, non-numbered paragraphs can be considered as located at the upper («zero») level. Therefore, to set or cancel paragraph numbering, it is possible to use the **Increase Nesting** and **Decrease Nesting** commands.

4.1.2.12.2. Setup of List Parameters

List parameters are the same for the whole text. Thus, uniformity is provided while processing all the lists contained in the document.



To set up list parameters, call the **List Parameters** command.

The dialog with the current settings for list parameters will be displayed on the screen (Fig. 4.1.21). The control elements of this dialog are represented in Table 4.1.14.

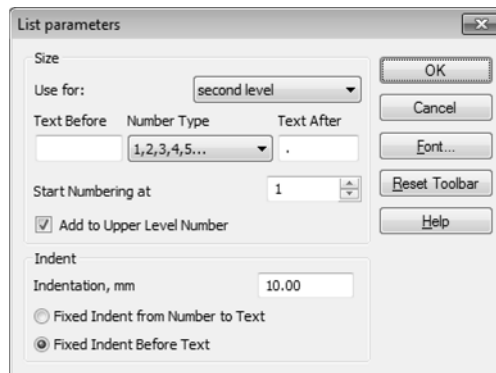


Fig. 4.1.21. List parameters dialog

Tab. 4.1.14. List parameters dialog

Element	Description
Use for	The list that allows to indicate the nesting level of the list, which is to be covered by formatting setup. The maximum number of levels is five [*] . If numbers of all levels are to be processed in the same way (Fig. 4.1.20, b), select the For All (by Default) option in this list. If level numbers are to be processed in different ways (Fig. 4.1.20 c, d), select levels in the list one by one and set up processing for each of them separately.
Text Before Text After	If these fields are filled in, their content is automatically added to list numbers. This capability is useful, for instance, while creating a numbered list of pictures (Fig. 4.1.22). To avoid duplication of the text from these fields while increasing the list nesting, it should be set only for those levels where it is required.
Start Numbering from Position	A field in which you can enter or select from the list a number from which numbering is to be started.
Add to Text from Upper Level	An option to manage forming of nested lists. If it is on, numbers of items at the given level include the number of the last item at the previous level, for instance, 2.2.1, 2.2.2. If the option is off, all the items at the given level have only their own serial numbers.
Indentation	Indent value in millimeters and its type (Table 4.1.15).
Font	A button that allows to set up parameters of the font used for numbers of list items. Once it is clicked, the font setup dialog will be displayed on the screen.
Reset	A button that allows to return to list parameters set by default.

* As practice shows, this is enough to process the overwhelming majority of documents. If nesting of the numbering list still exceeds five levels, numbers of the following levels will be processed in accordance with the setup made **For All (by Default)**.

Tab. 4.1.15. Types of indents in lists

Indent type	Description
Fixed Indent from Number to Text	The distance from the last symbol in the number up to the first symbol in the first line of each list item is constant and equal to the value preset in Indent Value field. The distance from the text border up to the first line symbol depends on how many symbols the number has.

Tab. 4.1.15. Types of indents in lists

Indent type	Description
Fixed Indent to Text Beginning	The distance from the text border up to the first symbol in the first line of each list item is constant and equal to the value preset in Indent Value field. The distance from the last symbol in the number up to the first line symbol depends on how many symbols the number has.

List of figures
Fig.1. Electromotor
Fig.2. Compressor
Fig.3. Gear box
Fig.4. Coupling
Fig.5. Control console
Fig.6. Load frame
Fig.7. Shock absorber

Fig. 4.1.22. Text before numbers of the first level

4.1.3. Spelling and Grammar Check

The spelling and grammar check program of KOMPAS-3D system serves to check spelling and grammar of text objects in KOMPAS documents:

- ▼ Text of a text document
- ▼ Text of BOM or Revision Table items,
- ▼ Title block text,
- ▼ Technical Specifications text,
- ▼ Text of inscriptions comprising an extension line, remote inscription, curly bracket,
- ▼ text on drawing,
- ▼ Text in tables.

Spelling and grammar check can be performed:

- ▼ In the entire document,
- ▼ In the currently edited text object.

Two checking methods are available:

- ▼ Automatic (see Section 4.1.3.2),
- ▼ Upon command call (see Section 4.1.3.3).

Checking by calling the command can be used either for the entire document or for the current edited text object.

Such text objects as an annotation text in a drawing, leader, remote inscription, curly bracket or table, can be selected before calling the command; then, the check will be run only on them. If no object is selected, then all text objects present in the document are checked.

When editing a certain text object, you can select a fragment before calling the command; then, only this fragment will be checked. If there are no selected fragments, then the whole text object will be checked.

Automatic check is available only for the text object being edited.

Spelling check in various KOMPAS documents is detailed in Section 4.1.3.5 on p. 1328.



To check spelling, remember that the program can find not all the errors, but only the most typical ones. Therefore, the final decision about errors being present or absent is left to the user.

4.1.3.1. Spell checking in different languages

A document or an object may contain texts in different languages. Their spelling will be checked as per rules and standards of the language attributed to them when entered or edited.

For details on how to enter text in different languages, see section 4.1.2.6 on p. 1301.

To enable spell checking, make sure that the dictionary file for the given language (**.lex*) is available in \Sys subfolder of KOMPAS-3D main directory.

To view the list of available language dictionaries, use the **Language** drop-down list in the **Formatting** tab of the Properties bar (see Fig. 4.1.3 on p. 1295).

To find in what language the document will be checked, select the text. The language will be shown in the **Language** field.

When the language field is empty, the selected text includes fragments from different languages. **No spelling check** means that the text does not have any attributed language and will be left non-checked.

The language attribute can be changed. E.g. when a text is pasted from the clipboard, the language may be unidentified. To enable spell checking, you should select the text and select the language in the drop-down list.

4.1.3.2. Automatic Text Check

An automatic check is done when editing a text object without the user intervention. The check command analyzes the text by finding the words, sentences or fragments suspected for errors.

The words suspected for errors are underlined by default with a red wavy line, the sentences — with a green line. Text with “No spell checking” attribute is underlined with a grey wavy line (underlining color can be adjusted, see Section 9.1.3.5 on p. 1681).

The automatic check can be enabled or disabled when setting up the program (see Section 4.1.3.4 on p. 1326).

To view the list of actions offered by the checking program to be applied to the word or sentence, call the context menu of the underlined element.

In the case of a spelling error, this menu allows:

- ▼ Replacing the word,
- ▼ Adding the word being checked to the dictionary,
- ▼ Not replacing and continuing the checking,

- ▼ selecting another spelling check language or “Do Not Check” attribute
In the case of a grammar error, this menu allows:
- ▼ Getting the error description
 - ▼ If the description is a direction, for example, “Comma required after the word “but” .”, then an automatic correction is possible according to the description,
 - ▼ if the description is a suggestion, for example, “Too many verbs in the sentence. Try...”, then no automatic correction is possible;
- ▼ Not correcting and continuing checking.

4.1.3.3. Spelling check upon Command Call



To start spelling check call the **Tools – Spelling...** command.

After that, the following steps will be sequentially performed for each sentence.

1. Spelling check of words.
2. Grammar check, if it is enabled (only for Russian language).

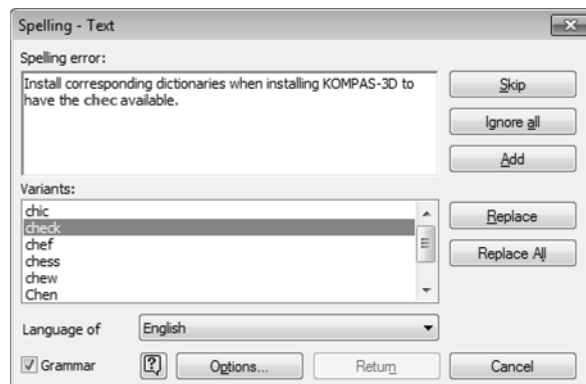


Fig. 4.1.23. Spelling check dialog

If a word is not found in the dictionary, as well as if analysis of the sentence allows to make a conclusion that there are grammar errors in it, the **Spelling** dialog is displayed on the screen (Fig. 4.1.23). The set of control elements of the **Spelling** dialog is given in Table 4.1.16.

Tab. 4.1.16. Spelling check control elements

Element	Description
Spelling Error, Grammar Error	<p>The field that contains the sentence being checked. During spelling check the word that is assumed to contain a spelling error is highlighted in red color. During grammar check part of the sentence that is assumed to contain a grammar error is highlighted in green color.</p> <p>The error can be corrected directly in this field. To enter corrections into the source text, click the Replace button during spelling check, and the Edit button during grammar check.</p> <p>When changing the option in the Dictionary language field, the content of the spelling error field changes.</p>
Variants	<p>During spell check this field contains the list of words that have spelling similar to that of the word being checked, from dictionaries of the spelling check program.</p> <p>The contents of the list are determined only by the spelling of words, therefore some words in the list may not be suitable in the given context.</p> <p>To replace the word being checked with a variant from the list, select it and click the Replace button.</p> <p>During grammar check this field contains brief descriptions of possible errors. To enter corrections into the original text, you need to select a variant and click the Edit button.</p> <p>Replacement or correction is also performed by double-clicking the list element. If you start editing of the word being checked in Spelling/Grammar Error field, Variants field becomes unavailable.</p>
Ignore	<p>A button that allows to skip the word (during spelling check) or sentence (during grammar check). It is considered verified and error-free. If you start editing the word/sentence being checked in Spelling /Grammar Error field, Ignore button is replaced with Undo Edit button.</p>
Undo Edit	<p>A button that allows to cancel editing performed in Spelling / Grammar Error field. Upon clicking the button, the field Variants becomes accessible.</p>

Tab. 4.1.16. Spelling check control elements

Element	Description
Ignore All	<p>A button that allows to skip the word (during spelling check) or sentence (during grammar check). It is considered verified and error-free. Words with the same spelling or sentences with the same grammar errors in the rest of the text are also considered correct and skipped during check (even if the highlighted fragment was checked).</p> <p>During spelling check the button becomes unavailable, if editing is performed in Spelling Error field.</p>
Add	<p>The button is present in the dialog during spelling check. The button serves to add the word being checked to an auxiliary dictionary that is set by default. During subsequent checking, this word will be offered as a replacement in the Variants list. The button is inaccessible when editing the Spelling Error field.</p>
Replace	<p>The button is present in the dialog during spelling check. The button serves to replace the checked word in the text with a variant from the list or with a word edited in the Spelling Error field. Upon clicking the button, the checking resumes.</p>
Change	<p>The button is present in the dialog during grammar check. It allows to edit the checked sentence in the text in accordance with the selected error alternative or replace it with the corrected one in Grammar Error field. Upon clicking the button, the checking resumes.</p>
Replace All	<p>The button is present in the dialog during spelling check. The button serves to replace the checked word and the words with similar spelling throughout the whole text with a variant from the list or with a word edited in the Spelling Error field. The already checked words, for example, those processed by the Ignore and Replace commands, will not be changed. Upon clicking the button, the checking resumes.</p>
Grammar	<p>The option serves to control grammar checking. By default, the option is enabled.</p>

Tab. 4.1.16. Spelling check control elements

Element	Description
Language of Dictionary	<p>The control allows you to change the language of the word being checked.</p> <p>The Language of Dictionary field displays the language attribute of the word, which is highlighted in red in the Spelling Error field. In the Variants field there is a list of replacement words from the language and auxiliary dictionaries.</p> <p>If necessary, select a desired option from the Language of Dictionary drop-down list.</p> <ul style="list-style-type: none"> ▼ If you select another dictionary language, the set of words in the Options field shall change. Select a word for replacement. ▼ If you select the Do Not Check option, this word shall remain unchecked. <p>When the check is complete, the checked words shall have the attributes of languages set in the dialog. The words, for which checking was cancelled, should not have a language attribute.</p>
Settings...	<p>The button serves to set up spell check options. Upon calling the command, the Spelling Options dialog will appear (see Section 4.1.3.4 on p. 1326).</p>
Restore	<p>The button serves to undo the previous text editing operation in the Spelling dialog. Upon undoing the last editing, the button becomes inaccessible. The button is also inaccessible before the first editing of the text.</p>
Cancel	<p>The button serves to finish spell check. Upon calling the command, the Spelling dialog will be closed, and the introduced changes will be saved in the text.</p>

4.1.3.4. Spell check Options Setup

Setting up spelling and grammar parameters is done in the **Spelling Options** dialog (Fig. 4.1.24). You can enter this dialog using one of two methods:

- ▼ Call the **Service — Options — System — Text Editor — Spelling Options** command.
- ▼ Click the **Options** button in the **Spelling** dialog.

Control elements of the spelling parameter setup dialog are given in Table 4.1.17.



Fig. 4.1.24. Dialog to set up spelling and grammar parameters

Tab. 4.1.17. Spelling check dialog

Element	Description
Spelling	A group of control elements to set up spell check.
Check Spelling Automatically	This option allows to enable automatic spell checking. The words suspected to have spelling errors will be underlined by default with a red wavy line.
Select only from the Basic Dictionary	The option allows using auxiliary dictionaries when checking spelling and grammar.
Ignore Words in UPPERCASE	The option serves to skip words in all capital letters during spell check.
Ignore Words with Numbers	The option serves to skip words combined of letters and digits during spell check.
Ignore Fractions, Indices and Superscripts	The option serves to skip fractions, indices, subscripts and superscripts during spell check.
Ignore Internet addresses and File Names	This option serves to skip Internet and e-mail addresses and file names during spell check.

Tab. 4.1.17. Spelling check dialog

Element	Description
Dictionaries...	The command allows to perform various operations with auxiliary dictionaries. Upon calling the command, the Additional Dictionaries dialog will appear (see Fig. 4.1.25 on p. 1330).
Grammar	A group of control elements to set up grammar check.
Check Grammar automatically	This option serves to enable automatic grammar check. Sentences or parts of sentences suspected to have errors will be underlined by default with a green wavy line.
Settings...	The button serves to set up the grammar parameters. Upon calling the command, the Spelling Settings dialog will appear.

*

4.1.3.5. Check Performance

Spell check in KOMPAS documents of various types has certain peculiarities. They are described in Sections 4.1.3.5.1 — 4.1.3.5.2.

4.1.3.5.1. Graphic al Document

Checking of spelling and grammar for all text objects in graphics documents.

The check starts after calling the **Service — Spelling** command. If an error is found in some text object, the **Spelling** dialog comes up on the screen (see Fig.4.1.16 on p. 1324 and Table 4.1.16). The display scale will change: the **Show All** command will be automatically applied to the document.

The name of the currently checked text object will be added to the dialog title. For example, when checking a text in a table, the dialog title will appear as **Spelling — Table**; when checking specifications — **Spelling — Technical specifications**.

Such text objects as text in a drawing, a table, an extension line, a curly bracket or a remote inscription are highlighted in the document during checking.

Upon closing the **Spelling** dialog, the document scale is restored.

4.1.3.5.2. Text Document

In text documents, the following items are checked:

- ▼ Text,
- ▼ Tables,
- ▼ Title block.

The inscriptions in the inserted fragments are not checked.

Texts in a basic inscription table are checked if they are visible on the screen, i.e., if the document is displayed in the page layout mode (see Section 4.1.5.1 on p. 1343). If the display of

design elements is switched off, i.e., when the document is displayed in the normal mode, only the text and tables of the document are checked.

The check starts after calling the **Service — Spelling** command. If an error is found, the respective text fragment is highlighted, and the **Spelling** dialog appears on the screen (see Fig.4.1.16 on p. 1324 and Table 4.1.16).

If a fragment (a portion of a text or a table) was selected in the document before calling the check command, then only that fragment is checked. Once checking is finished, a prompt to continue the remainder of the document is displayed.

If nothing was selected before calling the command, then the check will run from the current cursor position to the end of the document. After that, the beginning of the document is checked up to the cursor position at the time of calling the check. If the document is displayed in the print layout mode, then, after checking its contents (text and tables), the title block text is checked.

If necessary, the document is scrolled to display on the screen the text being currently checked.

To check the title block separately from the document contents, go into the title block editing mode and use automatic checking (see Section 4.1.3.2 on p. 1322) or checking upon command call (see Section 4.1.3.3 on p. 1323).

4.1.3.5.3. BOM

The following is checked in BOMs:

- ▼ Text of main and auxiliary items, including the text of the items not displayed in the table (items with identical text portion, items, whose display is manually disabled, twin objects),
- ▼ Title block.

Section titles and text in auxiliary columns is not checked.

To start checking, call the **Service — Spelling** command. If an error is found, the respective text fragment is highlighted, and the **Spelling** dialog appears on the screen (see Fig.4.1.16 on p. 1324 and Table 4.1.16).

If the BOM is displayed in the normal mode, then the BOM object text is checked. Note that:

- ▼ If none of items is being edited, then all items are checked,
- ▼ If a BOM item is being edited, then only that one will be checked:
 - ▼ If a text fragment was highlighted before the command call, only this fragment is checked; when the check is completed, the screen displays a request to continue the check in the rest of the BOM item.
 - ▼ If no fragments were selected, the entire text of the BOM item being edited will be checked.

If the BOM is displayed in the print layout mode, the text of BOM items is checked first, and then — the title block text.

To check the title block separately from the document contents, go into the title block editing mode and use automatic checking (see Section 4.1.3.2 on p. 1322) or checking upon command call (see Section 4.1.3.3 on p. 1323).

4.1.3.6. Dictionaries

Dictionaries are used during spell check. They contain the words that are confirmed to be correct. During a spell check, words in the text are compared to words in the dictionaries.

The delivery set includes the main dictionary with correct words (**.lex*) and the empty auxiliary dictionary (**.dic*).

The main dictionary contents are predefined and cannot be changed by KOMPAS-3D means.

The auxiliary dictionary is designed to add specialized technical terms and other special terms, proper nouns, abbreviations, etc., that are not included into the main dictionary.

The spelling and grammar check program allows creating additional auxiliary dictionaries. The auxiliary dictionary may include terms in different languages.

When a similarly spelled word is checked, the relevant term is searched automatically in all dictionaries as per the language and then proposed for substitution. If a word may belong to several languages (e.g. different parts of one term), then the system will propose the words from the first symbol language dictionary.

All the work with auxiliary dictionaries is performed in **Auxiliary Dictionaries** dialog (Fig. 4.1.25). It can be called using **Dictionaries...** button in the spelling setup dialog (see Section 4.1.3.4 on p. 1326).

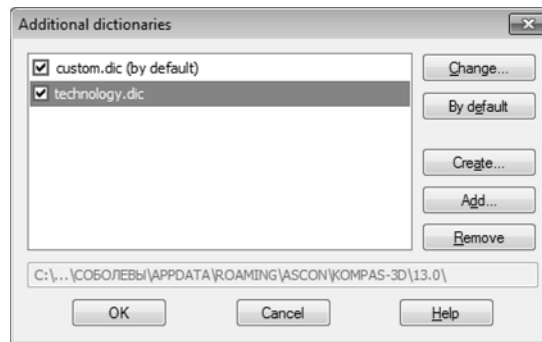


Fig. 4.1.25. The dialog to work with auxiliary dictionaries

4.1.3.6.1. Editing of the Auxiliary Dictionary

To edit an auxiliary dictionary, do the following.

1. Call the dialog to work with auxiliary dictionaries.
2. Highlight in the list the dictionary to be edited and click the **Edit** button.
3. In the dictionary editing dialog displayed on the screen (Fig.4.1.26) perform one of the following operations:
 - ▼ To add a word, enter it in the **Word:** field, and then click the **Add** button,
 - ▼ To delete a word, select it in the **Dictionary** field, and then click the **Delete** button,
 - ▼ to modify a word, delete it, and then add it with the changed spelling.

The language of a word added to the dictionary corresponds to the keyboard layout language.

4. Click the **Exit** button.

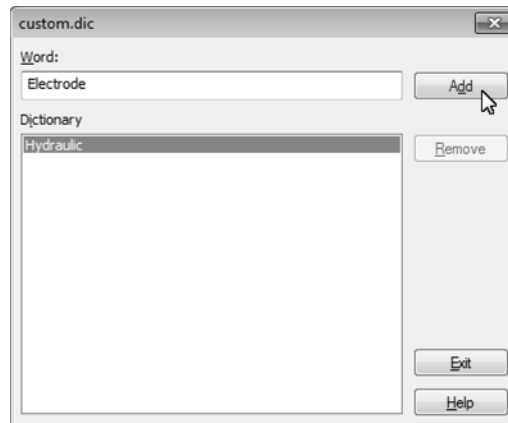


Fig. 4.1.26. The dialog for editing an auxiliary dictionary

4.1.3.6.2. New Auxiliary Dictionary Creating

To create an auxiliary dictionary, do the following.

1. Call the dialog to work with auxiliary dictionaries.
2. Click the **New** button.
3. In the dialog displayed on the screen, set the name and location for the dictionary file. Dictionary files are automatically assigned the extension **.dic*.
4. Click the **Save** button.

The file record dialog will be closed, and the name of the created dictionary will appear in the list of **Auxiliary Dictionaries** dialog. The option beside the dictionary name indicates whether it is used in the spelling and grammar check. By default, it is enabled.

A created dictionary is an empty file. Words will be added into it during spell check (see Section 4.1.3.6.4 on p. 1332).

If words are added during spell check, they are automatically introduced into a certain dictionary — the **default dictionary**.

5. To set a dictionary as the default, select its name in the list and click the **Default** button.
6. Click the **OK** button in the **Auxiliary Dictionaries** dialog.



To make the spell check command use additional dictionaries, disable the **Suggest from Main Dictionary Only** option in the Spelling and Grammar Settings dialog (see Table 4.1.17 on p. 1327).

4.1.3.6.3. Activating of Existing Auxiliary Dictionary

You can connect auxiliary dictionaries to the spelling and grammar checking system, for example, those provided by third-party developers. To connect an additional dictionary, do the following.

1. Call the dialog to work with auxiliary dictionaries.
2. Click the **Add** button.
3. In the dialog, select the desired dictionary file and click the **Open** button.
By default, an added dictionary is connected to the spelling and grammar checking system, i.e., the entry to the left of its name in the list of **Auxiliary Dictionaries** dialog is enabled.
4. To set a dictionary as the default, select its name in the list and click the **Default** button.
5. Click the **OK** button in the **Auxiliary Dictionaries** dialog.

4.1.3.6.4. Adding of Words to Auxiliary Dictionary during Spell check

To add the word being checked into the auxiliary dictionary during spelling and grammar check, click the **Add** button in **Spelling** dialog.

Added words are included into a certain auxiliary dictionary, i.e. default dictionary.

To check which of the auxiliary dictionaries is the default or to assign another dictionary as the default, do the following.

1. Click the **Options** button in the **Spelling** dialog (see Section 4.1.3.4 on p. 1326).
2. Click the **Dictionaries...** button in the **Spelling Options** dialog.
The **Auxiliary Dictionaries** dialog will appear on the screen.
The name of the default dictionary in the list is complemented with the “*(by default)*” line.
3. To set another dictionary as the default, select its name and click the **Default** button. Close the dialog by clicking the **OK** button.
4. Close the **Spelling Options** dialog.

4.1.4. Text in a Graphic Document

Text objects of a graphic document are:

- ▼ Inscriptions,
- ▼ Texts comprising complex objects (dimensions and designations).

You can set default parameters for the text in the drawing. To do this, call the **Service — Parameters... — Current Drawing (Fragment)**.

Select the **Default Font** item in the left part of the dialog that appears. From the list in the right part of the dialog select the font that will be used by default for all the inscriptions in the current document.

Select the **Text in the Drawing** item in the left part of the dialog. Control elements that appear in its right part allow to set up text parameters.

If you use the same text parameters in most graphic documents, it is not rational to perform the corresponding setup in every document. In this case, you can create all new documents with the required text parameters from the beginning.

To do this, call the **Service — Parameters... — New Documents — Graphical document** command.

Selecting the **Default Font** and **Text in the Drawing** items in the left part of the dialog, you can set up these parameters for all new graphic documents, as well as for the current document.

4.1.4.1. Inscriptions






To create a random text fragment in a drawing or in a fragment, call the **Tools — Text Input** command.

The Property toolbar will display control elements for locating an inscription. These elements are presented in Table 4.1.18.

Tab. 4.1.18. Control elements to locate an inscription in a drawing

Element	Description
Snap Point	Coordinate fields of the text anchor point.
Slope Angle	Slope angle of text lines to horizontal line.
Location	A group of switches that allow to select a method to locate the text relative to the snap point (Table 4.1.19).

Tab. 4.1.19. Methods to locate the text relative to the snap point

Method	Description
 Right	The text is located to the right of the snap point. This point will be aligned with the leftmost point of the first text line.
 Centered	The text is located symmetrically relative to the snap point. This point will be aligned with the center point of the first text line.
 Left	The text is located to the left of the snap point. This point will be aligned with the rightmost point of the first text line.

After the text snap point is indicated, the system switches to the text input mode: the screen displays the input frame, the structure of the Main menu is changed, and the **Format** and **Insert** tabs appear in the Property bar.

Enter the inscription text, pressing <Enter> key at the end of every line typed. By default, the size of the input frame is not fixed, and while typing it gets larger so that the text fully fits in the frame. If necessary, you can fix either width or height of the frame, as well as set the rules to locate the text inside the frame (see Section 4.1.4.1.2 on p. 1335).

While entering text in a drawing, you can use all the methods described in Section 4.1.2 on p. 1293, except for vertical insertion of text and illustrations.



To fix the inscription, click the **Create Object** button.



Upon calling the **Text Input** command once, you can create a few inscriptions with the same location parameters. After finishing inputting of the first text line, move the cursor outside the input box and click the left mouse button. The previous inscription will be automatically fixed, and a new text input frame will appear in the designated place.

4.1.4.1.1. Inscription Position and Text Editing

There are two methods to change the inscription position:

- ▼ Handles editing,
- ▼ Location parameters setup.

Unlike geometrical objects, an inscription has typical points of two types rather than of one. Their description is given in Table 4.1.20.

Tab. 4.1.20. Types of inscription handles

Cursor appearance when it is positioned on the handle	Purpose
	Inscription moving. The handle of the given type coincides with the snap point of the inscription.
	Rotation of the inscription around the snap point. The number of handles of the given type depends on the method to locate the inscription relative to the snap point (Table 4.1.19 on p. 1333).

Activation and moving of handles is performed by the common way (see Section 3.4.1.1.3 on p. 1114).

To set up inscription location parameters, do the following.

1. Enter the mode to edit the inscription. To do this, double-click it with the mouse.
2. Call the **Edit Location** context menu command. The Property bar will display control elements to locate the inscription (Table 4.1.18 on p. 1333).
3. Change the necessary inscription location parameters.
4. Confirm the performed changes by clicking the **Create Object** button.



To change the inscription text, enter its editing mode. Correct the text or change its parameters. Then confirm the performed changes.

In addition, the following editing commands can be applied to inscriptions:

- ▼ **Shift** (see Section 3.4.2 on p. 1120),
- ▼ **Rotation** (see Section 3.4.4.1 on p. 1129),
- ▼ **Symmetry** (see Section 3.4.4.3 on p. 1131)
- ▼ **Copying** (see Section 3.4.6 on p. 1136).

4.1.4.1.2. Drawing Inscriptions Format



To set location parameters and style for an inscription in a drawing or a fragment, call the **Formatting Options** command.

The text formatting dialog will be displayed on the screen (Fig. 4.1.27). Control elements for this dialog are given in Table 4.1.21.

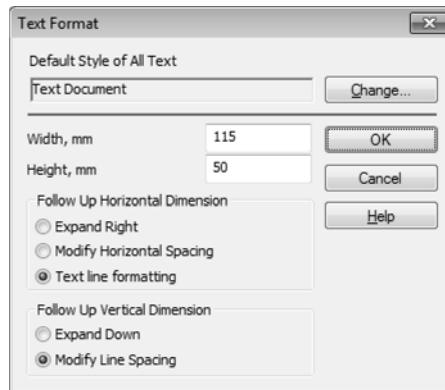


Fig. 4.1.27. Text formatting dialog

Tab. 4.1.21. Text formatting dialog

Element	Description
Default Style of All Text	A field that contains the name of the text style applied by default for inscriptions in the drawing.
Change	A button that allows to select a style for the current inscription. Once it is clicked, the screen will display the text style selection dialog (Table 4.1.7 on p. 1304). After assignment of another style, all the text of the current inscription will be reformatted in accordance with the new style. Other inscriptions in the drawing will remain unchanged. If different styles are required for different parts of the inscription, use the Style command (see Section 4.1.2.7.1 on p. 1302). Separate parts of the inscription can also be formatted using the Paragraph and Font (see Sections 4.1.2.5.1 on p. 1297 and 4.1.2.1 on p. 1294) commands.
Height, Width	Fields to enter overall dimensions for the rectangular area that limits the text (input frame). They are available while activating the Modify Horizontal Spacing (or Text Line Formatting) and Modify Line Spacing switches in the Follow Up Horizontal Dimension and Follow Up Vertical Dimension groups correspondingly.

Tab. 4.1.21. Text formatting dialog

Element	Description
Follow up Dimension Horizontal/Vertical	Groups of switches that allow to set the method to locate the text inside the input frame. These methods are detailed in Tables 4.1.22 and 4.1.23.

Tab. 4.1.22. Following up horizontal dimensions

Method	Description
Move right border	The right border of the input frame moves, tracking the length of entered lines. It is possible to move to a new line only by pressing <Enter> key.
Modify Horizontal Spacing	The right border of the input frame is rigidly fixed. Adjustment of line length to the frame width is performed by changing symbols narrowing. It is possible to move to a new line only by pressing <Enter> key.
Text Line Formatting	The right border of the input frame is rigidly fixed. Transfer of text from one line to another is performed automatically.

Tab. 4.1.23. Following up vertical dimensions

Method	Description
Expand Down	The lower border of the input frame moves, tracking vertical dimensions of the text fragment.
Edit Line Spacing	The lower border of the input frame is rigidly fixed. Adjustment of total text height to the frame height is performed by reducing the line spacing.

4.1.4.2. Texts Included into Complex Items

While entering inscriptions included into dimensions and designations, basic capabilities of text formatting and item insertion are available.

Formatting and insertion commands are performed by the common way (see Section 4.1.2 on p. 1293). Differences exist only in methods to access them.

While entering inscriptions included into dimensions and designations, insertion and formatting commands can be called using the same methods, as when entering inscriptions, i.e. using the Main menu, Tool panels and switches of the Properties bar.

While entering dimension inscriptions, access to insertion and formatting commands is performed using the **Insert** and **Format** sections in the menu of the inscription input dialog (see Fig. 3.3.4 on p. 980).

4.1.4.3. References

A **reference** is a text which is associatively related to another object containing a text. The object type with the link is called the link **source**.

When using the same source, the link contents may be different, i.e., text of the source, number of the page where it is located, etc.

Due to the presence of associative relation between the link and the source, the link contents is automatically updated when the source is changed.

For example, a tag symbol may contain a link to a specifications item number that defines the method and font to use for tagging. If the item number is changed (as a result of deleting or hiding items before it), the link will be updated. Due to this, the designation will always have the correct ID number of the technical specifications item.

KOMPAS-3D allows creating links between text objects within one graphic document.

Links can be included in the following text objects:

- ▼ technical specifications,
- ▼ text on drawing,
- ▼ table cells in a drawing,
- ▼ text in designation symbols,
- ▼ cells of the title block.

The following objects can be sources:

- ▼ layout objects (datum symbols, view arrows, reference numbers, etc.),
- ▼ text on drawing,
- ▼ View,
- ▼ Specifications item,
- ▼ Variable.
- ▼ document or object properties,
- ▼ associative report (including an array report with a table of modifiable variables), associative table of versions.

References on a drawing are highlighted with a color specified when setting up text elements (see Section 9.1.3.5 on p. 1681). Valid references are displayed by default in blue.

While connection between the link and its source is maintained, it is impossible to edit the link content: it always corresponds to the source text and its location zone designation. When deleting the source of a link, the relation between it and the link is broken. Links with broken connection are displayed in red color. A link can be automatically restored only by undoing the source object deletion using the **Edit — Undo** command. If the deletion can no longer be undone, then you will have to create the source object anew, and then edit the link again (see Section 4.1.4.3.4 on p. 1342).

If necessary, you can forcibly delete connection between the link and its source (see Section 4.1.4.3.5 on p. 1342).



The reference remains valid if its source is not shown in the screen. An example of such reference is a hidden projection designation or an object in a hidden view or a layer.

The colors specified in the text element settings are used only to display links located in the current fragment layer or in the current layer of the current drawing view. The links located on the other visible views and layers are not highlighted. Those are displayed in the same colors as the other objects on these views and layers. To print annotations that contain links, the colors defined by the print options are used.



As a result of creation of design objects in a drawing (sight arrows, profile/section lines and designations of extension elements), the views, designations of which contain links to the corresponding design objects, are automatically formed.

When creating and editing indicated design objects, you can include automatic adding of links to page number into designation of these objects and/or designation of the area where the corresponding view is located.

Associative relation between view designation and design object designation is detailed in Section 3.5.4.6.2 on p. 1166.

4.1.4.3.1. Reference Creation



To insert a link into the text, call the **Insert — Reference** command.

The screen will display the dialog shown in Fig.4.1.28. Setup reference parameters using the dialog's control elements (Table 4.1.24) and click **OK** to insert the reference into text.

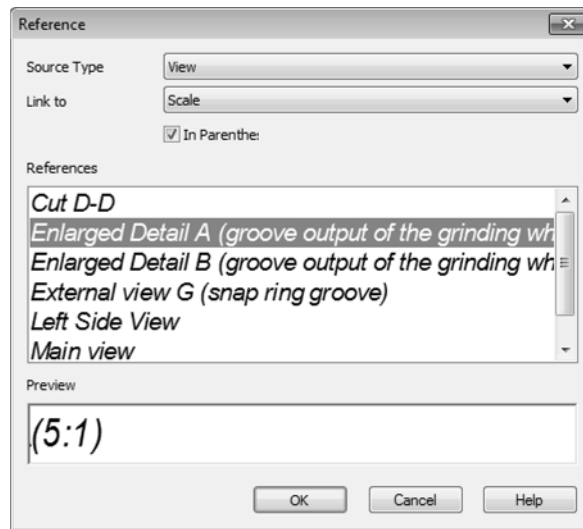


Fig. 4.1.28. Reference setup dialog

Tab. 4.1.24. Reference setup dialog

Element	Description
Source Type	This list serves to select the type of the link source object.

Tab. 4.1.24. Reference setup dialog

Element	Description
Link to	<p>This list serves to select the link contents.</p> <p>The set of link contents options depends on the source type. For most sources it is possible to put a link to a text, area designation or page number.</p> <p>If the source type belongs to View, creation of links has certain peculiarities (see Table 4.1.25 on p. 1340).</p> <p>If the source is located in several areas, then during creation of a link to area designation only one of areas is considered as source location (see Table 4.1.26 on p. 1341).</p>
In Parentheses	An option that allows to put the link content in parentheses.
Decimal Places	<p>The list appears in the dialog, if the Variable serves as a reference source and substitutes option In brackets.</p> <p>The list is available, if in list Reference to: the Value or Name = value option is selected.</p> <p>This list allows to specify number of decimal places to be displayed in a numerical value of a link to a Variable.</p> <p>The value of the value is rounded to the specified decimal.</p> <p>If you select 0, the link will display the whole rounded value.</p> <p>By default, the number of the digits after the decimal point will be the same as specified in digits setting dialog (see Section 9.1.3.5 on p. 1681). Maximum number is 6 characters.</p> <p>Angle values are presented in decimal system.</p>
List of sources	<p>This is the list displaying all texts of all sources of the selected type available in the document.</p> <p>The references are presented in sequence. A source, containing several text lines or shelves, which can be referenced, is displayed in the list as follows: the first line will go without indentation, like a single line source, and the second and subsequent lines will have an indentation at the left. For example, if one position designation contains several shelves with names, the reference to the position number will display all numbers in this designation — the first number without offset, while the rest are offset.</p> <p>To select the desired source, highlight its entry in the list. The selected source is also highlighted in the graphic document window (except for items of specifications).</p>
Preview	<p>This field displays the link being created just as it would appear when inserted in the text.</p> <p>If a link with specified parameters cannot be created (for example, when attempting to create a link to text under the shelf of the leader that contains text only above the shelf) the Preview field is empty and OK is inactive.</p>

4.1.4.3.2. Peculiarities of some types of links to objects

Drawing type

Peculiarities of creating link to a drawing view depend on the view type (associative or non-associative). Associative views are detailed in Part 3.6. Variants of links to the drawing's view are presented in Table 4.1.25.

Tab. 4.1.25. Options for the contents of the reference to a drawing view

Options for the link content	Description
Scale	Link contents is the current view scale.
Reoriented	<p>Link contents is the “reoriented” mark. A link can be created:</p> <ul style="list-style-type: none"> ▼ Always, if a view is nonassociative ▼ Under the following conditions, if the view is associative: <ul style="list-style-type: none"> ▼ For a cut/section and an enlarged element: if a sheet part flat pattern is depicted in the base view; ▼ For a view in arrow direction: if a sheet part flat pattern is depicted in it.
Rotated	<p>Link contents is the “rotated” mark. A link can be created:</p> <ul style="list-style-type: none"> ▼ Always, if a view is nonassociative ▼ Under the condition that the rotation angle defined on the Options tab is not zero[*], if a view is associative.
Rotated at Angle	The link contents are the current view rotation angle and the “rotated” icon. Conditions that allow to create a link are the same as for the previous option.

* If the view rotation is due to an existing projective relation to its base view (the **Projection Correlation** switch is active on the **Options** tab), then the link containing the rotation angle cannot be created.

Drawing zone

Creation of a link to the drawing's zone in which the selected link's source is located is possible, if zone splitting is enabled for the current drawing (see Section 3.5.8 on p. 1202). In fragment, zone splitting is impossible. Therefore, it is impossible to create a link to the zone in it.

The procedure of defining the link's contents for the cases, when the link's source is located in several drawing zones, is presented in Table 4.1.26.

Tab. 4.1.26. Defining the zone where the reference is located

Source Type	Location area is considered
Datum Symbol	Zone where the connection point of the frame and the leader line is located.
Cut Line	Zone where the snap point of the letter designating the first arrow of the view direction is located.
View Arrow	Zone where the snap point of the arrow-designating letter is located.
Brace	Snap point of a text; if the text is on the extension line, then it is the point where the line begins.
Text, Table of versions, Array report table, Report table	Zone where the text or table snap point is located.
Coordination Axis Mark	Zone where the mark is located. If both marks are enabled, then it is the zone containing the first mark.
Enlarged detail, Ref. No. Leader, Leader, Node Point Designation, Leader text, Sectional Node Point Designation, Mark/Reference Designation with Leader	Zone where the shelf start point is located.

Technical specifications

If specifications are the link's source, the link's contents may only be the number of the specifications' item. Therefore, to make link creation possible, the following conditions should be followed:

- ▼ each item of the specifications must be a separate paragraph,
- ▼ numbering for specifications' paragraphs should be set.



In order to number specifications' paragraphs, enter the respective edit mode, select the whole text and call the **Set numbering** command. For more information on the numbered lists, see Section 4.1.2.12 on p. 1317.



Specifications' paragraphs numbered manually (i.e. with numbers entered from the keyboard, not automatically) are in fact unnumbered. Therefore, creation of links to specifications' items is impossible in this case.



By default, the system is set up in such a way that the paragraphs of the specifications are created with numbers (see Section 9.2.6.18.6 on p. 1826).

4.1.4.3.3. Link updating

Updating a link means regenerating the contents of an existing link.

Updating a link is performed automatically upon the user's performing of any actions that change the text of the source object (for the links containing text) or the source object position (for the links containing the zone designations). Besides, all the links are unconditionally renewed when opening and closing the document, as well as when switching to the preview mode.

4.1.4.3.4. Link Editing

Link editing means changing its parameters. To edit a link, do the following.

1. Enter the editing mode of the text that contains the link.
2. Put the mouse cursor on the link and call the **Edit Link...** command from the context menu. The screen will display the same dialog as during creation of link (see Fig.4.1.28 on p. 1338 and Table 4.1.24).
3. Introduce the required changes and close the dialog by clicking the **OK** button.
4. Confirm text editing.

4.1.4.3.5. Link Explosion

Link explosion means deleting the relation between a link and its source. As a result of exploding, the contents of the link becomes a normal text that can be edited manually. An exploded link cannot be refreshed.

To explode a link, do the following.

1. Enter the editing mode of the text that contains the link.
2. Put the mouse cursor on the link and call the **Explode Link** command from the context menu.
3. Confirm text editing.

4.1.5. Creation of a Text Document

The file of the text document KOMPAS-3D has the *kdw* extension.

A text document can include one or several sections (see Section 4.1.5.3 on p. 1346).

Apart from the proper text, every page of a text document contains the title block. You can select the type of the title block for the first sheet, even and odd sheets. An integral characteristics of a text document is its format.

All these parameters can be set for a section (see Section 4.1.5.2 on p. 1343).

Sheet numbering in a text document and sections is detailed in Section 4.1.5.4 on p. 1346.

The title block of a text document is displayed or not displayed on the screen, depending on the document display mode.

4.1.5.1. Display Mode

There are two display modes for a text document:

- ▼ Normal,
- ▼ Page layout mode.



Switching between the normal mode and the page layout mode is performed using similar commands located in the **Service** menu. You can also use buttons in the **Modes** panel.

Right after creation, a text document is displayed in the Normal mode: in the document window, you see the white field with a dotted rectangular frame — the border of the text input area (Fig. 4.1.29, a).

In the page layout mode the screen also displays the frame and the title block of each page (Fig. 4.1.29, b). In other words, in the Page Layout mode the document is shown on the screen exactly as it will be printed on paper, including additional pages.

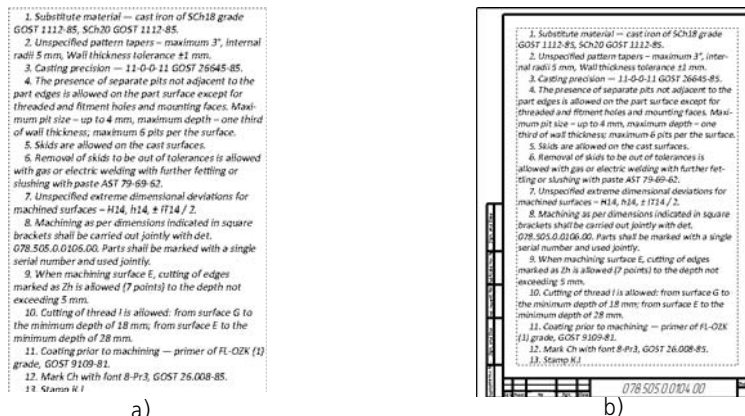


Fig. 4.1.29. Display of the same page of the document:
a) in the normal mode, b) in the page layout mode

The text on the document's main pages can be entered both in normal mode and in the layout mode; only layout mode is available for additional pages.

4.1.5.2. Title Block and Format

Title block is one of **layout** elements of a text document. Design also includes external and internal frames. Besides, design of the text document stores text field dimensions, i.e. distances from the internal frame to the text borders. Page designs are displayed on the screen only in the layout mode (see Section 4.1.5.1 on p. 1343).

LAYOUTS supplied with KOMPAS-3D are stored in libraries, i.e. in *.lyt files located in \Sys subfolder of the main system folder. The basic library of designs used when creating documents

is *graphic.lyt*. It is also possible to form user designs libraries. For details on text layouts, see Section 10.2.1 on p. 1959.

Due to the fact that each document page is assigned a design, you do not have to draw the frame and the table of the basic inscription.

By default, the first page of a text document created without a template (see Section 10.2.1.8 on p. 1982), has the design *Design text document. First Sheet. GOST 2.104-2006*. Even and odd pages have the design *Design text document. Next page. GOST 2.104-2006*. The default format of a text document is A4.

A text document created using a template has designs and the format of pages corresponding to the template.

Regardless of the method to create the document, you can modify designs and the format of its pages at any time.

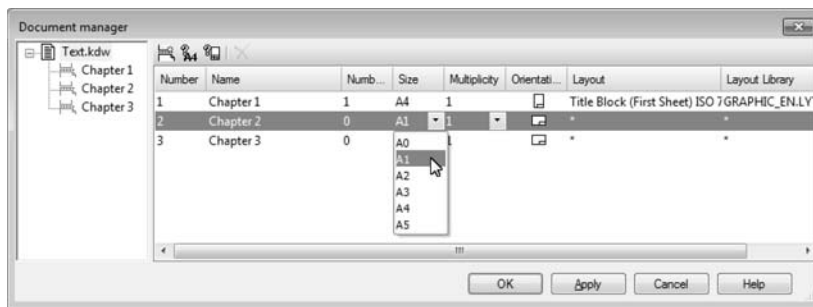


Fig. 4.1.30. Document manager; working with drawing sheets

To do this, perform the following actions.



1. Call the **Document manager** command. The **Document manager** dialog will be displayed (Fig. 4.1.30).

Sections are displayed in the document tree in the order of their creation. Properties of sheets within sections are shown to the right in relevant lines.

2. Click the line representing the section in the list to change the parameters need to be changed.
3. An icon in the **Orientation** column shows the current orientation of section sheets. To modify it, click the respective sign with the mouse. The icon will change as well.
4. The **Format** column shows the current format of section sheets. To change the format, open the drop-down list menu in this column and select an appropriate format.



You can set a non-standard sheet size. To do this, click the **Format** button on the **Document manager** toolbar. When a dialog box opens, set the **User** option, enter sheet size and close the dialog box. The **Format** column will show the selected sheet format.

5. The **Multiplicity** column will show the current sheet multiplicity. To change the multiplicity, open the drop-down list and select new value.



In case of a non-standard format, the multiplicity cannot be set.

6. The **Layout** column shows the name of the layout assigned to section sheets from the current layout library. This library filename is shown in the **Template Library**.

6.1. To select another template from current library, click the value in the **Template** column. Specify the required template in the dialog box that opens.



6.2. To select a template from another library, click the value in the **Template Library** column and click the **Template** button in the **Document Manager** toolbar. Specify the required library and template in the dialog box that will open.

If sheets inside one section have different layouts, “stars” are shown in the **Layout** and **Layout library** columns.

7. The changes can be viewed on screen without closing the dialog box. To do so, click **Apply**. To close the **Document Manager**, save changes and continue working with a drawing, click **OK**.

The layout and format of sheets in any section can also be changed by calling **Service — Parameters... — Current Text Document — Sheet Parameters**.

To set these sheet parameters for the specified section, select the **Format** and **Layout** in the left side of the dialog box. See Sections 9.2.4.2.1 on p. 1750 and 9.2.4.2.2 on p. 1750.



Layout is applied to section sheets. The first, even and odd sheet of the section is determined on the basis of the actual order of sheets. It does not depend on sheet numbers in the title block.

4.1.5.2.1. Filling in the title block

To fill in the title block of a text document, switch it to the layout mode and double-click the title block table with the mouse. The system will switch to the mode to fill in the basic inscription of the document. Methods to work in this mode are described in Sections 3.5.3.1.1–3.5.3.1.5.

After all title block cells have been filled in, click the **Create object** button or press <Ctrl>+<Enter> keys on the keyboard to save changes and exit the title block edit mode.



To exit from the mode to fill in the title block in a text document, you can click anywhere beyond the title block table.

The data entered into cells of the **Common for document** type is automatically transferred to other cells of the same type (this includes other sheets, as well). Cell types are preset during setup of title blocks included into page designs. For instance, in designs supplied together with the system, title blocks are set up in such a way that item designations entered into one of pages are transferred into *Designation* cells of title blocks on the rest pages.

Individual information is to be entered at each page.

Number of pages and *Page number* columns are filled in automatically and are not available for editing. You can set the predefined number of pages, as well as switch off automatic definition of page number (see Section 9.2.4.6 on p. 1753).

4.1.5.3. Sections

Section is a continuous part of a text document that can be different from the remaining document in terms of certain parameters. Sections are used to create sheets of a different format and orientation in one text document.

4.1.5.3.1. Adding and deleting a section

By default, a text document contains one section.

- ▼ A new section can be entered using two methods:



1. Go to **Insertion — Section break** or click the relevant button in the **Insert to text** toolbar. The new section will start at the current cursor position from the next page. It is separated from the previous one with the *Section break* caption. The new section has the format and layout of the previous or next section (if the new section is created in the beginning of the document).



2. Call the **Document manager** on the **Standard** bar. The Document Tree lists sections in the order they follow in the document, and lines in the right part of the **Document manager** show properties of sheets within sections.



- 2.1. On the **Document manager** toolbar, click **Create section**. A new section appears in the end of the list with default parameters. If any of the existing sections was selected before the button is clicked, the new section is inserted after the selected one and has the same parameters.

If necessary, change the section's name and properties of its sheets (see Section 4.1.5.2 on p. 1343).

- 2.2. Click the **OK** button in the dialog **Document Manager**. It will close and the added section will appear in the text document.

- ▼ To delete a section, call the **Document manager**, select the section to be deleted in the Tree and click **Delete** in the toolbar.



A text document cannot exist without sections. Thus, it is impossible to delete the last sheet in a document.

4.1.5.3.2. Switching between sections

The **Current section** field of the **Current status** bar shows the name and number (in parentheses) of the document's current section. To switch to another section, expand the list to the right of the **Current section** field and indicate the necessary line in the list. The screen will show the first page of the required section.

4.1.5.4. Numbering

A text document supports automatic numbering of sheets. If it is enabled, the number of sheets and their numbers shown in the *Number of sheets* and *Sheet number* columns of the title block are determined automatically and depend on the setting.

- ▼ Number in the *Number of sheets* column:

- ▼ can be preset (set by the user at setup),
- ▼ can be determined automatically as the sum ¹of sheets of all sections.

- ▼ Number in the *Sheet number* column.

Number of the first sheet in the section:

- ▼ can be preset (set by the user at setup),
- ▼ can be determined by the numbering of the previous section; in this case, the number is calculated as $S + 1$, where S — the total¹ number of sheets in previous sections (this variant is unavailable for the document's first section).

Numbers of the second and subsequent sheets are calculated as $N + 1$, where N is the number of the previous sheet.

If auto numbering is enabled for all sections and starts with 1, the number of each sheet corresponds to the actual number of the document's sheet.

Individual sections can be excluded from numbering. Sheets from these sections will not be included in the *Sheet number* and *Number of sheets* columns.

When automatic numbering is disabled, numbers and the number of sheets can be manually entered in the *Sheet number* and *Number of sheets* columns.

Numbering parameters for the current document are setup in the dialog called with **Service — Parameters... — Current text document— Sheet numbering**. (see Section 9.2.4.6 on p. 1753).

1. the sum includes:
— the specified number of the section's sheets if this number is predefined;
— the actual number of the section's sheets if this number is defined automatically.

4.2. Tables

4.2.1. General Information

KOMPAS-3D Text Editor offers a multitude of options to create and edit various tables in both graphic and text documents.

The techniques for working with tables are generally the same. The main commands for working with tables are grouped together in **Table** menu, and buttons for these commands can be found in the **Tables and Borders** toolbar (Fig.4.2.1).

In addition, commands can be called using the option buttons located on the **Table** tab of the Properties bar (Fig.4.2.2).



Fig. 4.2.1. **Tables and Borders** toolbar



Fig. 4.2.2. **Table** tab

When entering text into table cells, you can use all formatting techniques described in Section 4.1.2 on p. 1293. Formatting parameters may also be specified for an empty cell.

An existing table can be transformed by adding or removing columns and rows, splitting or merging cells, resizing cell borders and changing their style.

Created standard table (i.e., gear parameters table) can be saved in a separate file and then inserted into new documents.

4.2.1.1. Working Techniques

All working techniques described below are available in the table editing mode.

The system automatically switches to the table editing mode once the new table has been created.

To switch to editing of an existing table, double-click it.

<i>Designation</i>	<i>Gear ring</i>	<i>b</i>	<i>d1</i>	<i>n</i>	<i>i</i>
894 1409 0022 008	A	4	14.5	4	1:6
-01	B	-	-	3	1:3
-02	C	-	-	3	1:1

Fig. 4.2.3. Table in editing mode

In the editing mode, a thin border is drawn around the table, and text fields within the table cells are shown in dashed lines (Fig.4.2.3). The cursor remains in the cell or position where it was before double-clicking.

4.2.1.1.1. Entering Text into Table Cell

Entering text into a table cell is similar to entering the normal text.

To switch to the desired cell, click it with the left mouse button. In addition, you can navigate cells using <Tab>.

The cell containing cursor is called **current cell**. The column and the row which the cell belongs to are also called **current**.

4.2.1.1.2. Selecting Table Cells, Rows and Columns

Selecting table cells, rows or columns may be useful, for example, for editing text in corresponding rows or columns.

A range of neighboring cells can be selected in two ways.

- ▼ Move your cursor across the table while holding down the left mouse button.
- ▼ Place the cursor in the corner cell of the range to be selected, then press and hold *Shift* and left-click the opposite corner cell.

<i>Designation</i>	<i>Layer</i>	<i>View</i>
<i>078.505.9.0200.00</i>		<i>Base</i>
<i>078.505.9.0200.01</i>		<i>Export</i>
<i>078.505.9.0200.02</i>		<i>Tropic</i>

Fig. 4.2.4. Cell range selection example



There are two ways of selecting a table row/column.

- ▼ **By using menu commands.** Place the cursor in any cell of the row/column to be selected and use **Select Row (Select Column)** command.
- ▼ **Using the mouse.** Move the mouse pointer to the table border against the row/column to be selected. When the mouse pointer will change its shape to an arrow directed towards the table, click the left mouse button. This will select the row/column which the arrow is pointing at.



There are two ways of selecting a range of table rows/columns.

- ▼ Move the mouse pointer to the table border against the first row/column of the range to be selected. When the mouse pointer will change its shape to an arrow directed towards the table, click and hold the left mouse button and move the pointer to the last row/column of the range to be selected. The selection will cover all rows/columns between the starting and ending row/column of the range. When all the necessary rows/columns will be selected, release the left mouse button.
- ▼ Select the first row/column of the range to be selected, then press and hold *Shift* and left-click any cell of the last row/column of the range.



To select a cell, move the mouse pointer to its left border. When the mouse pointer will change its shape to an arrow directed towards the cell, click the left mouse button. This will select the cell which the arrow is pointing at.

4.2.1.1.3. Merging Cells



To merge a range of cells into one cell, select the range and use **Merge Cells** command.

The command will only be available if all selected cells are adjacent and form a rectangular selection.

Styles of cell borders being merged may vary. Border style of a merged cell is selected according to the following rules:

- ▼ The style of top horizontal border corresponds to the style of top border of the upper left cell in the selected set.
- ▼ The style of bottom horizontal border corresponds to the style of bottom border of the lower left cell in the selected set.
- ▼ The style of left vertical border corresponds to the style of left border of the top left cell in The selected set.
- ▼ The style of right vertical border corresponds to the style of right border of the top right cell in the selected set.

Rules for creating text in merged cells:

- ▼ Texts of cells being merged are added to the text of the top left cell according to their location order, i.e., left to right and top to bottom.
- ▼ Texts of cells being merged retain the following parameters of paragraph: *row step, break line, indents, tabulation, intervals, aligning*;
- ▼ Texts of cells being merged save the following parameters of the font: *Font, height, spacing, Italic, Bold, Underlined, Color*;
- ▼ The text of every merged cell starts from a new paragraph in the merged cell.
- ▼ Cells containing no text do not create an empty paragraph.
- ▼ Values of *Offsets as well as check horizontal size, and check vertical size* parameters are equal to corresponding values of top left cell parameters.

4.2.1.1.4. Splitting Cells



To split the current cell or a range of selected table cells into several cells, use **Split Cells** command.

A **Splitting Cells** dialog will appear (Fig.4.2.5). You will be prompted to specify the number of rows and columns which the cell(s) will be split into.

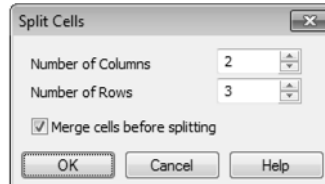


Fig. 4.2.5. Splitting cells dialog

If the command is applied to a set of selected cells, its result depends on the **Merge Cells before Splitting** option of this dialog.

- ▼ If the option is enabled then selected cells are merged and this created cell is splitted into specified number of rows and columns.
- ▼ If the option is disabled, then every selected cell is splitted into specified number of rows and columns.



If selected cells form a non-rectangular shape, they cannot be merged; therefore, **Merge Cells before Splitting** option will be unavailable. This option will also be unavailable when splitting a single cell.

Cell's properties and its format are inherited from source cell. If cells were merged before splitting, formatting properties and settings will be inherited from the upper left cell of the range.

4.2.1.1.5. Adding or Removing Rows and Columns

To insert a row/column, use the appropriate command:



- ▼ **Insert Column to the Left,**



- ▼ **Insert Column to the Right,**



- ▼ **Insert Row above,**



- ▼ **Insert Row below.**

The row/column that was the current row/column before calling the command will be the **source** row/column for the new row/column.



To add a line at the bottom of the table, set the cursor in the end position of the bottom right cell and press the `<Tab>` key.

The rules for inserting are as follows.

- ▼ One row/column will be inserted per each insert command.
- ▼ The cells of the new row/column will have the same size, text formatting settings and border style as the source row/column.
- ▼ The text contained in the source row/column cells will not be copied to the new row/column cells.



To remove rows or columns, use **Delete Column** or **Delete Row** commands.



To delete more than one row/column at once, select them before calling this command.



Pressing the *Delete* key deletes the contents of cells in the selected columns (rows). The columns (rows) themselves remain in the table.



You can only insert rows/columns if the number of rows/columns is not fixed (See Section 4.2.1.1.10 on p. 1357).

4.2.1.1.6. Copying and Pasting Cells, Rows and Columns

Copying or moving cells, rows and columns between tables is possible within the same document as well as between different documents opened in the same KOMPAS-3D application (including different document types).



In contrast to copying or moving text, copying or moving cells, rows or columns is performed using KOMPAS-3D's own clipboard.

The cells, rows or columns are copied and moved with maintaining the font settings, paragraph settings and text style.

You copy or move cells, rows or columns from one table to another by doing the following.

1. Enter table editing mode for the table, from which the cells, rows or columns will be copied or moved.



2. Select the desired range of cells, rows or columns.

3. Use context menu item **Copy** to copy the range to clipboard or use **Cut** to cut.



4. Enter table editing mode for the table into which the cells, rows or columns will be pasted.

5. Place the cursor where you want them to be pasted:

- ▼ When pasting cells— in the cell where the contents of the upper left cell of the range in the clipboard should be pasted.
- ▼ When pasting rows— in the row above which the rows in the clipboard should be pasted.
- ▼ When pasting columns— in the column to the left of which the columns in the clipboard should be pasted.



6. Use context menu item **Paste**.

The cells in the clipboard will be pasted into the table to the right and below of the current cell. The contents of the pasted cells will replace the contents of the target cells. If necessary, additional rows/columns will be added to the table.

For correct row pasting, it is required that both tables have the same number of columns, and for correct column pasting, that they have the same number of rows. Otherwise, pasting may become incomplete, or merged cells can appear.

If table dimensions are blocked (see Section 4.2.1.1.10 on p. 1357) or restriction to modify the text or to enter multiline text has been enabled for the cells (see Section 4.2.1.1.11 on p. 1357), then a dialog prompting to remove the restrictions may appear upon issuing the paste command. To proceed with pasting, click **Yes**, otherwise click **No**.

4.2.1.1.7. Resizing Table Cells

You resize a cell by doing the following.



1. Move the mouse pointer to the cell border. It will change to a two-way arrow.
2. Click and hold the left mouse button and drag the cell border as desired.
3. Once the desired border is reached, release the mouse button.



By moving the pointer to the cell corner (intersection of cell borders), you can drag both cell borders at the same time. The mouse pointer in this case will look like a four-way arrow.

Dragging the cell borders will change, in general, the size of cells and tables in the following way.

- ▼ The respective dimension of all cells of the current column or row will change in accordance with the mouse pointer position.
- ▼ The table dimensions remain unchanged.
- ▼ Dimension of the column to the right from the current one and/or the lower row will change in accordance with the mouse pointer position.

If, when dragging the cell border, button *Ctrl* is pressed, the sizes change as follows.

- ▼ The respective dimension of all cells of the current column or row will change in accordance with the mouse pointer position.
- ▼ Dimensions of neighboring columns and rows remain unchanged.
- ▼ The table sizes will change with respect to the cell new sizes.

During dragging, current cell dimensions will be displayed next to the pointer (Fig.4.2.6).

<i>Designation</i>	<i>Ring gear</i>	<i>i</i>
<i>894 1409 0022 009</i>	<i>A</i>	<i>1.6</i>
<i>-01</i>	<i>B</i>	<i>1.3</i>
<i>-02</i>	<i>C</i>	<i>1.1</i>

Fig. 4.2.6. Resizing cell borders

- ▼ When resizing either horizontal or vertical cell borders, the height or the width of the cell will be displayed. The border being resized and its parallel dimension measuring border will be highlighted.

- ▼ When resizing the cell corner, cell height and width will be displayed at the same time. The borders being resized and their parallel dimension measuring borders will be highlighted.

Please note that the dimensions displayed next to the pointer pertain to the cell to the left or above the border being resized, except when resizing the upper or right border of the table. In this case, the dimensions displayed next to the pointer will pertain to the cell located to the right or below the border being resized.



To know the cell dimensions, move the pointer to its lower right corner. When the pointer turns into a four-way arrow, left-click with the mouse. Cell dimensions will appear next to the pointer.

Cell dimensions can be changed with the mouse and are displayed next to the pointer with an accuracy of 1mm. To resize the border with greater accuracy, enter the desired dimension in the cell format settings dialog (see Section 4.2.1.1.11 on p. 1357).



Resizing row or column borders is only possible if cell dimensions are not fixed (see Section 4.2.1.1.10 on p. 1357).

4.2.1.1.8. Cell Borders

The cells of the newly created table will have default borders: external borders and lines separating table rows will have *Normal* style and column separating lines will have *Thin* style.



Manage cell border display and style using border settings dialog (Fig. 4.2.7). Bring it up using **Borders** command.

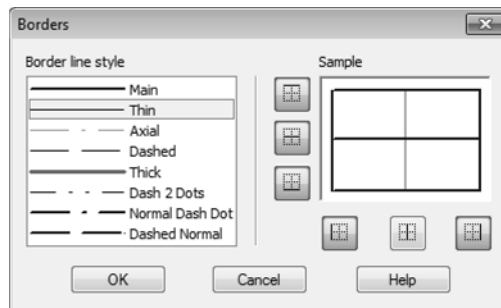


Fig. 4.2.7. Dialog for Setting up Borders

The **Border Line Style** list itemizes the line styles that can be used for cell borders.

The **Sample** field schematically displays the selected range of cells. To the left and below the field there are buttons corresponding to the borders. If the border display is enabled, then the button is pressed, if disabled— released. The enabled boundaries are displayed on the sample by their specified line style, while the disabled ones— in gray lines. If the boundary of the selected range consists of various-style lines then it is shown on the sample as a solid gray line.

To disable the display of the current cell borders (or of the selected cells range), release the buttons corresponding to those borders in the border settings dialog.



For convenience of editing a table with disabled borders, you can turn on their display on the screen by activating the switch **Show Grid**. The disabled borders will be shown as thin gray lines. These lines are displayed only in the mode of table editing and will not be printed.

Change the border line style of the current cell (or selected cell range) by doing the following.

1. Bring up the border settings dialog (Fig.4.2.7).
2. Select the desired style from the **Border Line Style** list.
3. If the border, whose style needs to be changed, is disabled, click its respective button. If the border, whose style needs to be changed, is enabled, release its respective button, and then click it again.

You can also control the appearance of borders using the buttons on the **Tables and Borders** toolbar.

To change the border line style of the current cell (or selected cell range), select the desired style from the list and then click the button corresponding to the desired border location (Fig.4.2.8).

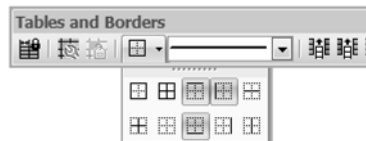


Fig. 4.2.8. Selecting cell border line style



The buttons for individual borders can be grouped on a separate toolbar that can be positioned in any convenient location. To do this, drag the button menu with the mouse at its title in any direction. The **Borders** toolbar will be created.



If the **Tables and Borders** panel is positioned vertical, it will show a **Line Style** button instead of the style list. Click this button to access the list of border line styles. A dialog will appear allowing you to choose the desired style.

4.2.1.1.9. Saving Tables

Once a table is created, it may be saved to a file for further use. In KOMPAS-3D, table files have *tbl* extension.

You save the current table to an external file by doing the following.



1. Call the **Save Table to File** command.
2. A dialog will appear prompting for a directory and file name to save the table.

Once the table is saved, you can open it from the disk. To do this, click **From File** button in the table creation dialog (see Table. 4.2.6 on p. 1361).

4.2.1.1.10. Fixing Table Size



When creating template tables, you can prevent from changing their structure and cell sizes. You do this using **Fix Table Size** command.

A lock settings dialog will be displayed (Fig.4.2.9). The settings in this dialog are described in the Table4.2.1.

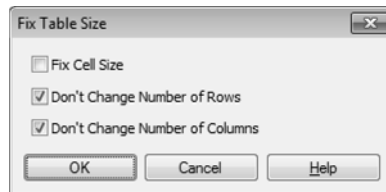


Fig. 4.2.9. Fix Table Size dialog

Tab. 4.2.1. Fix Table Size dialog

Option	Description
Fix Cell Size	Prevents from changing cell dimensions.
Prohibit to change the number of rows	Disables paste row and delete row commands.
Prohibit to change number of columns	Disables paste column and delete column commands.

4.2.1.1.11. Formatting Cells



You can adjust the formatting of the current cell or selected cells using **Cell Format** command.

A cell format dialog will appear (Fig.4.2.10). The control elements of this dialog are represented in Table 4.2.2.

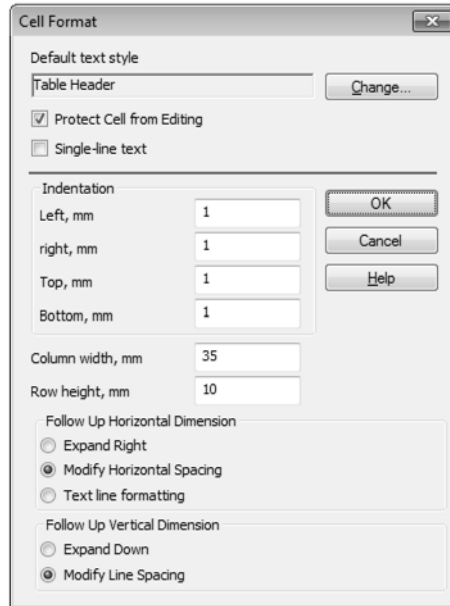


Fig. 4.2.10. Cell format dialog

Tab. 4.2.2. Cell format dialog

Element	Description
Default Text Style	A field containing the name of the text style used by default for the text in the cell.
Change	A button allowing to select the text style for the selected cell(s). Once it is clicked, the screen will display the text style selection dialog (Table 4.1.7 on p. 1304). If another style is selected, all text in the cell(s) will be formatted according to the new style. Other cells will not be affected.
Prohibit Protect Cell from Editing	An option that makes it possible to prevent all editing of the selected cell(s) contents. Such a restriction can be useful, i.e., for making template tables that have cells with text that should not be changed.
Single-Line Text	An option that prevents from entering more than one line of text into the cell. Like the previous option, it is used for making template tables.
Indentations	An option group determining text layout in the cell. Indents are measured from cell borders. It is not recommended to set zero indents because this will cause text characters to touch the cell borders.

Tab. 4.2.2. Cell format dialog

Element	Description
Column Width, Row Height	Here you can enter the dimensions of the current cell. They are available while activating the Modify Horizontal Spacing (or Text Line Formatting) and Modify Line Spacing switches in the Follow Up Horizontal Dimension and Follow Up Vertical Dimension groups correspondingly.
Follow up Dimension Horizontal/Vertical	Option groups allowing to set the way of laying out the text in the cell(s). These methods are detailed in Tables 4.2.3 and 4.2.4.

Tab. 4.2.3. Following up horizontal dimensions

Method	Description
Move right border	Length of text lines in the cell will be dynamically monitored: if a text line longer than the cell width is entered, the right cell border (and the right border of the entire column) will be resized to the right. All columns to the right from the current column will move further to the right. It is possible to move to a new line only by pressing <Enter> key.
Modify Horizontal Spacing	Cell (and the whole column) width will remain constant during text input*. Line length is matched to cell width by adjusting the horizontal character spacing. It is possible to move to a new line only by pressing <Enter> key.
Text Line Formatting	Cell (and the whole column) width will remain constant during text input*. Line length is matched to cell width by automatically wrapping the text to the next line.

* The cell could still be resized manually (see Section 4.2.1.1.7 on p. 1354).

Tab. 4.2.4. Following up vertical dimensions

Method	Description
Expand Down	The vertical dimension of the text in the cell will be dynamically monitored: if the text does not fit to cell, the cell lower border (and the border of the whole row) will be resized down.
Edit Line Spacing	Cell (and the whole row) height will remain constant during text input. The overall text height fits to cell height by adjusting the line spacing.



If the cell size is fixed (see Section 4.2.1.1.10 on p. 1357), the indent fields and the **Expand Right** and **Expand Down** options will be unavailable.

4.2.1.2. Setting up Table Text Parameters

The text in table heading cells usually differs from the text in other cells in character size, alignment and other settings.

When a new table is created, the user specifies the row in which the heading will be (see Table). 4.2.6 on p. 1361).

The text in the heading cells will use *Table Header* system style, while all other cells will use *Table Cell* style.

If necessary, you can adjust the settings of these system styles in the current graphic or text document. To do this, call the **Service — Parameters... — Current Document**.

You will see **Table Header** and **Table Cell** items in the list of setup objects (in the left-hand portion of the dialog that appears). When selected, control elements will be displayed in the right-hand portion of the dialog (see Table 9.2.4 on p. 1752) allowing to set up the respective system styles for the text in the current document.

When the dialog is closed, the text with system styles *Table Header* and *Table Cell* will be reformatted.

The setting will be saved **in the current document and will not change when opening it on another workplace**.

If most documents are to use *Table Header* and *Table Cell* styles with similar settings, it may be inefficient to adjust the relevant settings in each document individually. In this case, you can create all new documents with all desired settings already in place. To do this, call the **Service — Parameters... — New documents**.

In the left-hand portion of the dialog that appears, open the section corresponding to the document type being configured: either **Text Document** or **Graphic Document**. Then, configure the *Table Header* and *Table Cell* styles by selecting the relevant items.



Service— Options... command will be unavailable if the system is in table editing mode. Therefore, before configuring, you will need to exit this mode by left-clicking outside the table being edited.

4.2.2. Using Tables

Pasting tables in graphic and text documents is done almost identically.

To edit the pasted table afterwards, double-click it with the mouse.

4.2.2.1. Tables in Graphic Documents



To paste a table into a drawing or fragment, use **Tools— Add Table** command.

The Properties bar will display control elements for table placement. These elements are presented in Table 4.2.5.

Tab. 4.2.5. Table layout control elements

Element	Description
Snap Point	Coordinate fields of the table anchor point.
Slope Angle	Here you can enter the angle of incline of table rows off horizontal.

After specifying the table anchor point, a new table dialog will appear (Fig.4.2.11). The control elements of this dialog are represented in Table 4.2.6.

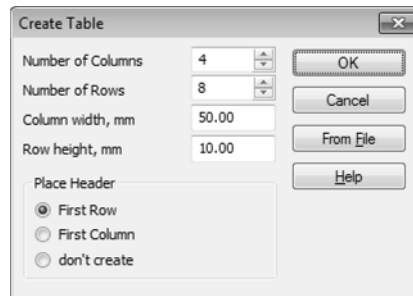


Fig. 4.2.11. Dialog for creating a new table

Tab. 4.2.6. Dialog for creating a new table

Element	Description
Number of Columns, Number of Rows	Here you can enter the number of rows and columns of the new table.
Column Width, Line Height	Here you can enter cell dimensions of the new table.
Place Header	An option group controlling creation and positioning of table heading. The most common example is a table with parameter names as a heading and parameter values in the table cells.
From File	A button that allows to insert a ready table from an external file (see Section 4.2.2.1.1). Once clicked, a menu will appear prompting for the type of the file that has the desired table. When inserting a table from a file, the table structure settings made in this dialog will be ignored.

The newly created table or the table generated from the fragment (see Section 4.2.2.1.1) will be placed in the document so that its upper left corner is at the anchor point.



During table editing, its upper left corner can be moved as necessary, but the anchor point position will remain constant.

When a table is read from a *.tbl* file (see *ibid*), it will be placed in the document so that its anchor point has the specified coordinates.

The cursor will automatically be placed in the upper left cell of the new table. You can input text in cells, edit and format it as well as change the table structure.



To lock the table, click **Create Object** button.

4.2.2.1.1. Inserting Table from File

The opportunity to insert tables from external files allows to use template tables repeatedly once they have been created.

Tables are inserted into a document from a file by embedding. Embedding allows you not to worry about the transferring the file from which the table was inserted when transferring the document to other workplaces.

Tab. 4.2.7. Commands for inserting tables from external files

Command	Description
Load from a Table File	Loading tables saved in <i>.tbl</i> files (KOMPAS-3D table file) *. Upon calling the command, a dialog in which you can select the desired file <i>.tbl</i> is displayed.
New from a Graphic Fragment	<p>Creating tables from <i>.frw</i> files (KOMPAS-3D fragment file). Upon calling the command, a dialog in which you can select the <i>.frw</i> file containing the table form is displayed. Once a fragment has been selected and opened, a table creation settings dialog will appear (see Fig.4.2.12), where you can adjust certain parameters of the table being generated.</p> <p>When automatically generating a table from a graphic fragment, all its text as well as its horizontal and vertical line segments will be accounted for. Other objects are not involved in building up the table. Styles of text and line segments are saved in the resulting table (in the way of text styles within the cells and the styles of cell borders).</p>

* About saving tables to *.tbl* file: see Section 4.2.1.1.9 on p. 1356.

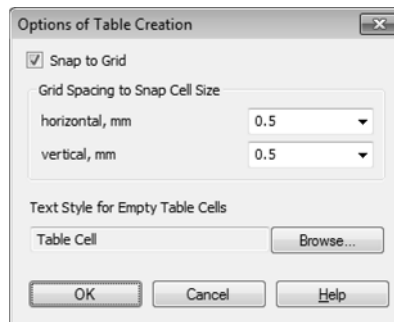


Fig. 4.2.12. Options of Table Creation dialog

Tab. 4.2.8. Options of Table Creation dialog

Element	Description
Use Snap to Grid	If this option is enabled, cell dimensions will be stepped (as multiples of a particular number). Dimensions of table cells in the fragment will be rounded to the nearest value that is a multiple of the specified grid step. If this option is disabled, cell dimensions of the resulting table will be the same as the cell dimensions of the table in the fragment.
Grid Spacing to Snap Cell Size	Field group for entering the horizontal and vertical step. The group will only be available if cell dimension snapping to grid is enabled.
Text Style for Empty Table Cells	Text style that will be used by default when entering text in table cells that were empty in the fragment*.
Edit...	A button for assigning another style for empty cells. When clicked, a text style selection dialog will appear (see Table). 4.1.7 on p. 1304).

* Text style of those cells that were not empty will not be affected.

4.2.2.1.2. Editing Table

Table position can be changed in two ways:

- ▼ Handles editing,
- ▼ Location parameters setup.

Editing characteristic points of the table is done in a regular fashion (see Section 3.4.1.1.3 on p. 1114).

Adjust table position settings by doing the following.

1. Enter the mode to edit the inscription. To do this, double-click it with the mouse.
2. Call the **Edit Location** context menu command. Table layout controls will appear on the Properties bar (Table). 4.2.5 on p. 1361).
3. Adjust the anchor point position and/or table tilt angle.



4. Confirm the adjustments by clicking **Create Object** button.

To adjust the table structure or the text in its cells, switch to table editing mode. Modify the table as necessary and confirm changes.

In addition, you can apply the following edit commands to tables:

- ▼ **Shift** (see Section 3.4.2 on p. 1120),
- ▼ **Rotation** (see Section 3.4.4.1 on p. 1129),
- ▼ **Symmetry** (see Section 3.4.4.3 on p. 1131)
- ▼ **Copying** (see Section 3.4.3 on p. 1122).

You can also copy and transfer tables using a clipboard (see Section 3.1.3.5 on p. 882).

4.2.2.1.3. Table names and numbering

In a graphical document, a table can have a manually entered name and/or the number set automatically. The name and the number are entered independently. Table name entry is described in this section; automatic object numbering is described in Section 3.9.1 on p. 1277.

In order to assign a name to a table, call the **Name...** command from the table's context menu. After the command is called, the screen displays the table name entry dialog. If a table, for example, an associative report is split into several parts, the dialog contains text entry fields, the number of which is equal to the number of tables (Fig. 4.2.13).

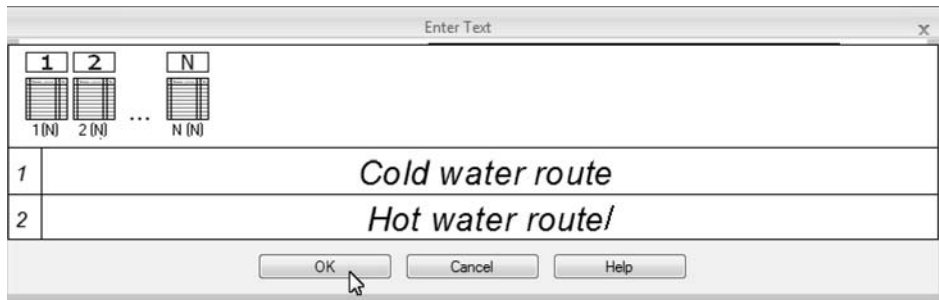


Fig. 4.2.13. The dialog for entry of the associative table name

Enter the name of each table one by one in the dialog's fields.

The line's length cannot be more than the table's width. In this case, automatic shrinking of symbols or text wrapping will not take place. In order to split a name into lines, use the `<Enter>` button; to go to the name of another table, use `<⌘/⌥≠LM>` keys.

Elements are displayed on the Properties toolbar and commands serving to work with the text are available in the Main menu to work with text. Some of them can be called from the context menu of the dialog's input field. For instance, you can enter a special character or a symbol, or change text formatting parameters: font, height, leveling and indentation units, etc.

Once you have finished name entry or editing, click the **OK** button in the dialog. The dialog will be closed, and the newly set or changed names will appear above the tables.

To reject input or editing of names, click the **Cancel** button.

You can adjust name text parameters in the current document:

- ▼ for simple tables — in the dialog which is called by the following command: **Service — Parameters — Current drawing/fragment — Table parameters — Table name.**
- ▼ for associative tables — in the dialog which is called by the following command: **Service — Parameters — Current drawing/fragment — Table parameters — Table name.**

If automatic numbering was applied to the current table, the number (it may also include the text in front of the number) is displayed in the dialog. Such number is the reference to the table number in the numbering group and is not editable. If necessary, you can delete the reference number. In this case, the automatic numbering of the current table will be canceled.

4.2.2.2. Tables in Text Document



To insert a table into a text document, use **Insert— Table...** command.

A new table dialog will appear (see Fig.4.2.11 on p. 1361).

The table will be inserted in the text as one paragraph.

If there was any text in the line before the cursor, a new empty paragraph will be created automatically to accommodate the table.

The cursor will automatically be placed in the upper left cell of the new table. You can input text in cells, edit and format it as well as change the table structure.



To lock the table, click **Create Object** button.

To change the layout of the table with respect to the text area margins and the spacing between the table and the previous and next paragraphs, configure its host paragraph (see Section 4.1.2.5.1 on p. 1297).



Fitting to width for the table will be equivalent to aligning left.

Copying and moving tables via the clipboard is the same as copying and moving text fragments (see Section 4.1.2.4 on p. 1296).

4.2.2.3. Other Cases of Using Tables

The table editing process takes place not only when an arbitrary table is inserted in the document, but also when a geometric tolerance symbol is created (see Section 3.3.7.13 on p. 1045), and also when generating a title block table. In all cases mentioned above, all basic techniques and methods of table management are available.

5. Properties and reports

5.1. Properties

5.1.1. General information

Property – information associated with a graphic image or a model reflecting a certain feature of them, for example, designation, name, mass, etc.

The document property list is created and stored in a drawing, in a fragment or in a model document. It is common for a document and for objects contained in it while values of the object's properties may differ.

Properties can be set:

- ▼ in graphic documents— for a drawing or a whole fragment, macro elements, view and fragment inserts;
- ▼ in a model document — for the entire model, bodies and components.

In the associative view of the drawing, properties of the model, for which the drawing has been prepared, can be available for viewing.

Document properties are divided into the *system* and *advanced* ones. By default, the property list contains system properties only. You can add complementary properties to the list.

While working, you can edit the list of complementary properties: create and add new properties, modify and delete the existing ones, add properties from property libraries. The outlined actions are described in Section 5.1.2.5 on p. 1375.

You can preset a property list for new documents by including all necessary properties in it. Use property libraries for this. Connection of libraries is described in Section 5.1.2.1. The KOM-PAS-3D system is delivered with the property library *properties.lpt*.

You can create your own property libraries (see Section 5.1.2.4 on p. 1374).

To set the property value, the property from the list should be reflected in the document's properties. This setting is applicable both to new documents (see Section 5.1.2.1) and to the current one (see Section 5.1.2.2 on p. 1372).

Property values are assigned while you work with the document. Setting of property values is described in Section 5.1.4 on p. 1384.

Properties of the document and of its objects can be used for representation in the form of reports. See Section 5.2 on p. 1403 for more details on reports.

5.1.2. Property Management

The property list is customized for new documents or for the current one.

When setting up new documents, you can connect necessary property libraries and enable the display of properties in a document, whereas setting up the current document— enable the display of properties in it only.

5.1.2.1. Customizing property list of new documents

In order to change the factor for graphic documents, call the **Service — Parameters... — New documents — Graphic document — Property list setup** for new model documents, call the following command: **Service — Parameters... — New Documents — Model — Part/Assembly — Property list setup** command.

The screen will display the dialog shown in Fig.5.1.1.

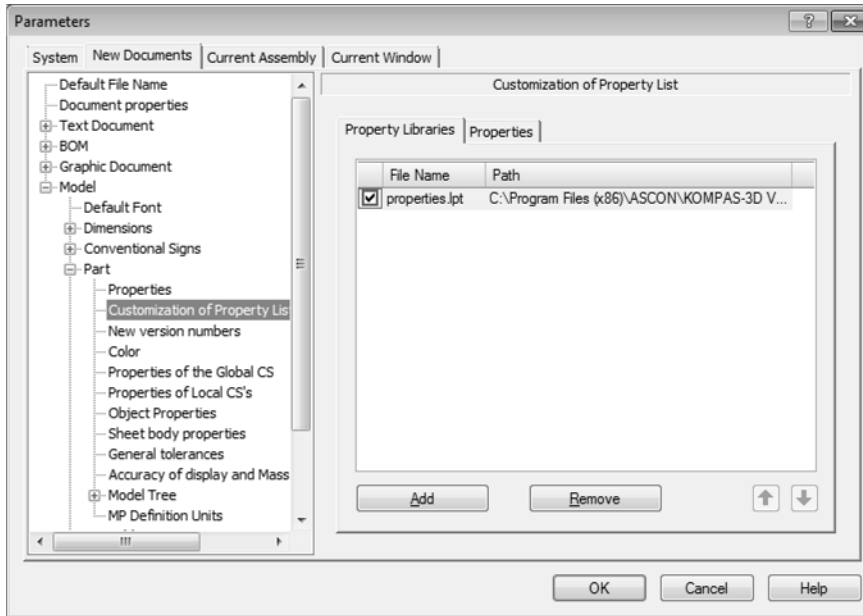


Fig. 5.1.1. Customizing property list of new documents, connecting property library

The dialog has two tabs: **Property Libraries** and **Properties**.

The **Property Libraries** tab allows to connect the libraries which properties can be used in new documents.

To connect a library, do the following.

1. Add the property library to the library list. For this, click the **Add** button and select the required library (the *.lpt* file) in the Windows standard file open dialog. The library list will include the row containing library file name and its path.



2. Enable the option to the left of the library file name in the list. Library properties will be added to the property list of new documents.



You can edit the library list by moving or deleting list items. Use the **Move Up/Down** and **Delete** buttons for this.

The property list of new documents is on the **Properties** tab. Activate this tab to customize display of properties in new documents (Fig.5.1.2).

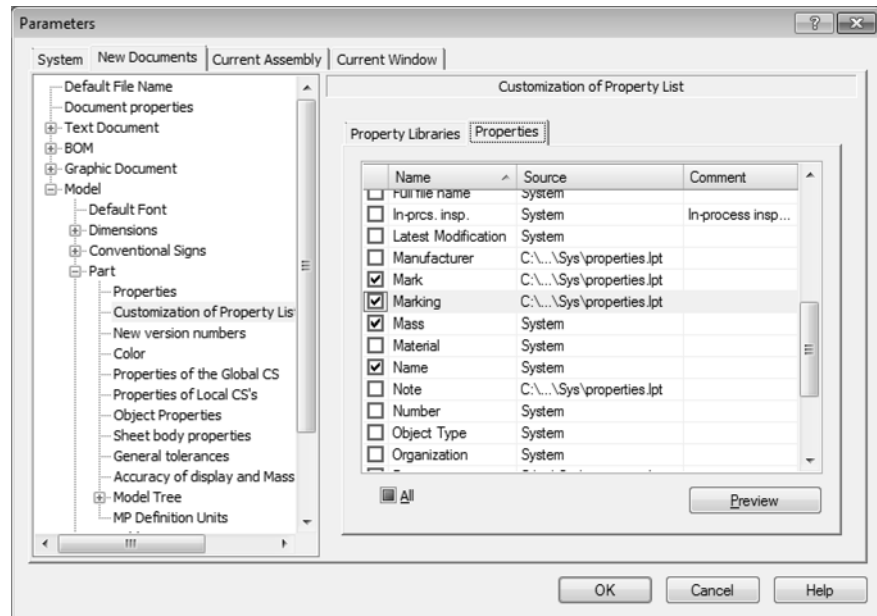


Fig. 5.1.2. Customizing property list of new documents, customizing display of properties

The list includes system properties and properties from connected libraries. The option to the left of the property name allows to show or hide this property in new documents.

The **All** option allows to show/hide all properties.

Click the **View** button to view parameters of the property selected in the list. The **Property Parameters** dialog will be displayed (Fig.5.1.3). In the view mode the data contained in the dialog are uneditable.

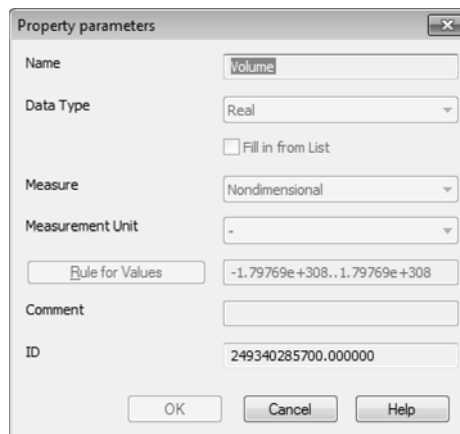


Fig. 5.1.3. Viewing property parameters

5.1.2.2. Customizing property list of current document

In order to set up the list of properties for the current document (drawing, fragment, part or assembly), call the following command: **Service — Parameters... — Current document — Property list setup**. The screen will display the dialog shown in Fig.5.1.4.

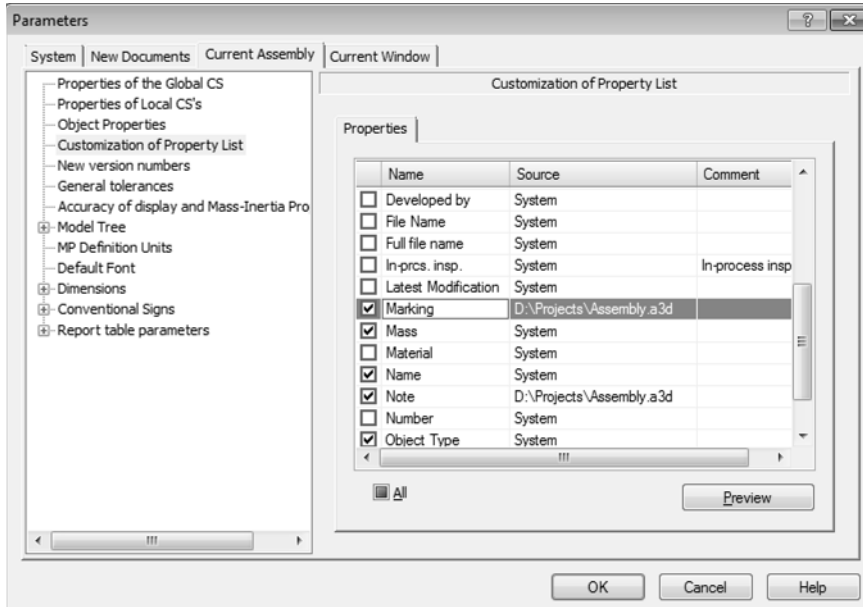


Fig. 5.1.4. Customizing property list of current document

The dialog contains one tab **Properties**. This tab contains the property list of the current document. Aside from system properties, the list may contain properties set by the user and properties added from connected libraries and files of the document's objects, for example, components or view and fragment insertions when working with a drawing (see Section 5.1.2.3). Customize displaying of properties in the current document by enabling or disabling options to the left of property names.

5.1.2.3. Property Management



You can work with document properties and property libraries in the **Property Management** dialog (Fig.5.1.5) that is called with the command **Service— Libraries of Styles, Layout Types and Title Blocks— Object Properties...**. The dialog can also be called from the property bar during creation of properties for a document or its objects, as well as in the Property Editor. More information about setting properties and the Editor can be found in Section 5.1.4 on p. 1384.

In this dialog, you can create but also enable the display of a property in the current document.

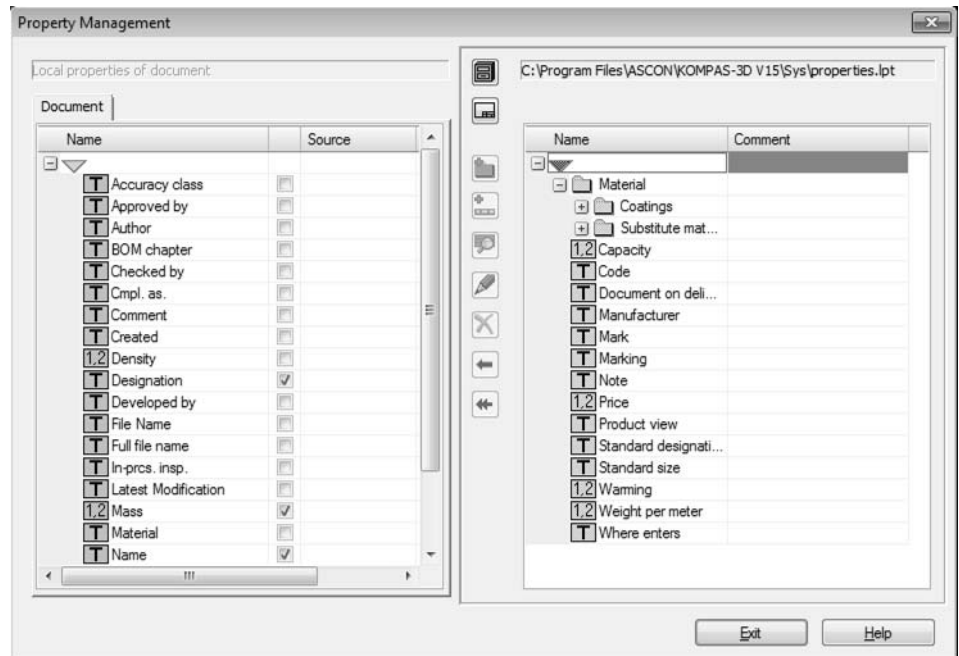


Fig. 5.1.5. Assembly Property Management dialog

The dialog contains two preview windows. To perform actions in one of the windows, activate it by clicking it. And the buttons located between the windows become available for work in an active window.

Only one window can be active it is selected by a marquee. For example, in Fig. 5.1.5, the right window is active.

Every view window allows to display the property list of the current document or of the chosen property library.



Document properties can be displayed with the **Show Document** button and Property Library can be displayed with the **Show Library** button.



Figure 5.1.5 shows the left view window displaying the **Local Document Property** (properties of the current document) list and the right view window displaying the property list from the *properties.lpt* library.

View window appearance depends on the document type chosen to display properties.

- ▼ For a part, the view window contains the **Document** tab that displays the property list of the current document.
- ▼ For an assembly, drawing or fragment the view window contains two tabs This **Document** tab displaying the property list of the current document. For an assembly, the **Components** tab displays the list of first level attributes for the components of the assembly. For a drawing or a fragment on the **Insertions** tab— the list of view/fragments insertions' attributes. There are only properties different from document properties on **Components** or **Insert** tabs.



In the **Insertions** tab of the drawing, you can also find properties of the model for which the associative view is created.

The property list on the **Document** tab appears as a table. The source and commentary are displayed in the table for each property, as well as the option for its connection to the document. The source, in this case, may be the System, library of properties or a document from which the property is added.

Enabled option means that this property is displayed in a document.

- ▼ The view window contains no tabs for the property library. The library property list is displayed in the window field. In the library groups of properties can be created at different levels. The property list appears in the window as a tree, containing sections and subsections.

Every list property has a name and an icon. Icon characters show the property value type:

- ▼ the *T* character corresponds to the value type *Row*,
- ▼ the *L* character corresponds to the value type *Logical*,
- ▼ the *1,2* characters correspond to the value type *Integral* or *Real-valued*.

The system property icon is colored in gray, the complementary property icon is colored in yellow.



To view property information, select it in the list and click the **View** button. The **Property Parameters** dialog will be displayed (see Fig. 5.1.3 on p. 1371). In the view mode the data contained in the dialog are uneditable.

You can create new property libraries (see Section 5.1.2.4), add properties to the property list of the current document or library, modify or delete the existing properties (see Section 5.1.2.5 on p. 1375).

5.1.2.4. Creating property library

The property library is created in the **Property Management** dialog (see Fig.5.1.5 on p. 1373).

Do the following to create a library.



1. Click the **Show Library** button.
2. In the dialog that appears on the screen, select or create a folder in which a new library will be located.
3. Enter a name of a new (non-existing) library and click the **Open** button.
4. Respond **Yes** to a system prompt for creation of a new file (Fig.5.1.6).

The active preview window of a property management dialog will be cleared since the newly created library is empty.

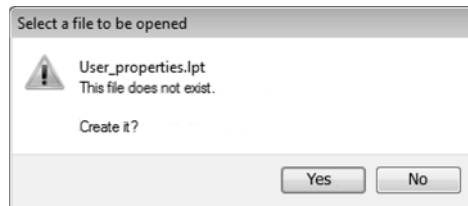


Fig. 5.1.6. Prompt for creation of a new library file



5. To create a new section in a library, click the **Create Section** button and enter the section name in the displayed dialog. Create all required sections and subsections in a similar way.



6. Create necessary properties using the **Create Property** button. Property creation steps are detailed in Section 5.1.2.5.1 on p. 1375.

When needed, you can edit and delete library sections and properties (see Sections 5.1.2.5.2 and 5.1.2.5.3 on p. 1379).



The **Copy** button allows to copy the existing properties to the library



The **Transfer** button allows to transfer them.



To do this, open the desired property source in the adjacent view window to copy (transfer) and select the property.



The source can be the current document or a property library.

- ▼ When selecting a *document*, you can not only copy (transfer) properties of this document but also copy properties of insertions that are part of it.
- ▼ When selecting a *library*, you can copy (transfer) both properties and sections of this library.



System document properties cannot be copied and transferred.

5.1.2.5. Generating document property list

By default, the document property list contains system properties and properties from the libraries connected during customization (see Section 5.1.2.1 on p. 1370). You can edit the list by creating new properties, modifying or deleting the existing ones. The list is edited in the **Property Management** dialog (see Fig.5.1.5 on p. 1373).

5.1.2.5.1. Property creation



Show the property list of the current document by clicking the **Show Document** button. The view window will open the **Document** tab containing this list.



Click the **Create Property** button to add a new property to the list. The **Property Parameters** dialog will be displayed (Fig. 5.1.7). The elements of this dialog are described in Table 5.1.1.

Fig. 5.1.7. Property Creation Dialog

Tab. 5.1.1. Property Creation Dialog

Element	Description
Name	The field allows to enter property name. If the property with such name already exists in the list, the name of the newly created property is complemented with a sequence number.
Data Type	The property value type can be selected in the dropdown list— <i>Integer</i> , <i>Real</i> , <i>String</i> or <i>Logic</i> . The selected value type determines whether the dialog controls will be available.
Get from List	The option allows to use a list of predefined values when a property value is assigned. To generate the list, enable this option and click the Rule for Values button. The option is available for all property value types except for the logical one.
Value	The drop-down list allows to choose the measurable entity to which the property will relate. Once the measurable entity is chosen, its basic unit of measurement becomes property unit of measurement. The list is available for the <i>Real</i> or <i>String</i> property value types.
Unit	The drop-down list allows to select property unit of measurement. It contains units of measurement of the current measurable entity. The list is available for the <i>Real</i> or <i>String</i> property value types.

Tab. 5.1.1. Property Creation Dialog

Element	Description
Rule for Values	<p>Property values can be defined within a particular value range or selected from the predefined list (when the Get from the List option is enabled). The first variant is available for values with <i>Integer</i> or <i>Real</i> types, and the second one— for values with <i>Integer</i>, <i>Real</i> or <i>String</i> types.</p> <p>When you assign the value range, the Value Input Rule dialog will pop up after the button is pressed (Fig. 5.1.8). It contains minimum and maximum property value input fields.</p> <p>When you assign the value list, the Value List dialog will pop up after the button is pressed (Fig.5.1.9). The dialog controls are described in Table 5.1.2 on p. 1378.</p>
Value Field	This field is intended for displaying a rule defined for the property values filling in. It is not available for editing.
Comment	This field allows to input additional information for a property. It is available for values of all types.
ID	This field is intended for displaying the property ID. The ID is assigned automatically and cannot be modified by the user. The field is non-editable.

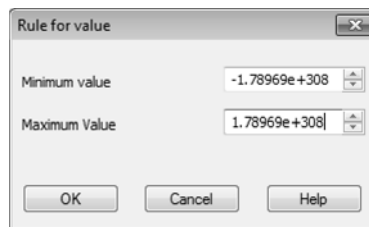


Fig. 5.1.8. Property Value Range Assignment Dialog

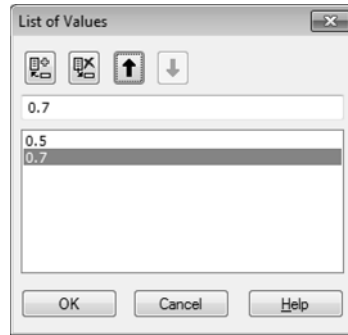






Fig. 5.1.9. Property Value List Creation Dialog

Tab. 5.1.2. Property Value List Creation Dialog

Name	Description
Value Input Field	This field serves for a new value inputting. To add the input value to the list, click the Add Value button or press <i><Enter></i> .
List Displaying Field	The field serves to display the generated value list.
 Add value	This button allows to add a value defined in the input field to the list of values.
 Delete Value	The button allows to delete the value selected in the list.
 Move the Value up/down in the List	The button allows to move the value selected in the list one position up or down.
	

The newly created property is added to the property list of the current document. It is shown automatically.

5.1.2.5.2. Property Modification



In the **Property Management** dialog (see Fig.5.1.5 on p. 1373) enable the displaying of the property list of the current document by clicking the **Show Document** button. The view window will open the **Document** tab containing this list.



To modify a property of the current document, choose this property from the property list and press the **Edit** button. The **Property Parameters** dialog will be displayed (see Fig. 5.1.7 on p. 1376). The dialog controls are described in Table 5.1.1 on p. 1376

While editing a property, you can modify the property name and its additional data, choose a unit of measurement from the list of units of measurement of the current measurable entity, modify limit values of the specified value range or edit the generated property value list.

A property value type, measurable entity and value input rule are assigned when a property is created and cannot be modified.



Changes in system property parameters are saved during the current working session of KOMPAS-3D.

5.1.2.5.3. Property Deletion



In the **Property Management** dialog (see Fig.5.1.5 on p. 1373) enable the displaying of the property list of the current document by clicking the **Show Document** button. The view window will open the **Document** tab containing this list.



To delete a property, select it in the property list and press the **Delete** button. The request to confirm deletion will pop up. If the property being deleted has the value assigned in the current document, the delete request will contain information about this. Click the **Yes** button to confirm deletion or the **No** button to reject it.

Note that you can delete a complementary document property only. System properties cannot be deleted.



You cannot undo property deletion.

5.1.2.5.4. Adding property from property library to document

Do the following to add a property from a library to a document.



1. Enable displaying of the property list of the current document in one of the **Property Management** (see Fig.5.1.5 on p. 1373) dialog view windows. To do this, click the **Show Document** button. The window will open the **Document** tab containing the property list.



2. Show the library property list in the other dialog window. For this, click the **Show Library** button and choose the required library file from the Windows standard file open dialog.



3. Copy or transfer the property from the library to the document.



To copy the property, select it in the library property list and press the **Copy Right/Left** button.



To transfer the selected property, click the **Transfer Right/Left** button. The property will be deleted from the property library and added to the document property list.



After a property is transferred from the library to the local property list, the property is saved to the document even if the library is deleted.



You can use a ready *properties.lpt* library delivered with the KOMPAS-3D system.

5.1.2.5.5. Copying of properties of a component (or insertion) to a document-assembly (or drawing/fragment)

Assembly (or drawing/fragment insertions) component properties can be copied to the property list of this document. For this, do the following.



1. Enable displaying of the property list of the current document in one of the **Property Management** (see Fig.5.1.5 on p. 1373) dialog view windows. To do this, click the **Show Document** button. The window will open the **Document** tab containing the property list.



2. Show the component property list of the current assembly (or a drawing/fragment insertions) in the other dialog window. To do this, click the **Show Document** button and open the **Components** (or **Insertions** tab).



The **Components** (or **Insertions**) tab contains the **component (or insertion) properties different from those of the current document**. These properties are available only for viewing and copying.



3. Select the required property in the list of properties of components (or insertions) and click the **Cop to the right/left** button. The component property will be copied to the current document property list.



5.1.2.5.6. Properties connecting to a document

Connecting a property to a document is required for setting a value for the property.

To connect the property, enable the option near the property on the **Document** tab of the **Local Document Properties** window in the **Properties Management** dialog.

Connected properties are displayed in the list when setting the properties' values (see Section 5.1.4 on p. 1384).

5.1.3. Property types and values

5.1.3.1. System properties

System properties are properties which are always present in a document and can be used in it, for example, *Designation*, *Name*, *Mass* and so on. Values of some system properties are set automatically – based on data obtained by the system when working with a certain object. Other values are manually set by the user in the Property bar, Property Editor, or using other methods.

▼ **Designation***, **Name***

Values are set manually or by a link to another object in the document (see Section 5.1.4.4.1 on p. 1395). In a drawing, values can be set in the title block.



You can choose document or objects included in it designation from the external Reference (Classifier of the Unified system for design documentation). Use the **Select Designation from Reference** button for this. The reference must be connected to KOMPAS-3D.



Values of properties marked with an asterisk (*) can be set in the Property Editor (see Section 5.1.4.2 on p. 1389).

▼ Quantity

This property value is set to one since you can currently work with properties of one document or object only.

▼ Mass*

In graphic documents the value is set manually. In a drawing, the value can be set in the title block. By default, in properties of an associative drawing, you can see the mass specified in the model based on which the view is created.

The mass value of a model or its part is obtained when mass-inertia properties are calculated or assigned manually on the **MP Parameters** tab of the Properties Bar (see Section 2.14.6 on p. 802). It is automatically transferred to the document property list.

▼ Material*, Replacement for material*, Coating*, Density*

In the graphic document, values are set in the **Properties** tab of the Properties bar. They can be entered manually or selected from references. In a drawing, the material can also be set in the title block.

In the model, material is selected from the reference, while density, replacements for material and coatings can be selected from the reference or entered manually. Material of the model or its part can be set in the **Properties** tab or the **MP Parameters** tab of the Properties bar. When material is selected, its designation, density and hatch style are transferred to the document's properties list.

In order to select a material, a replacement for material and coating, you can use buttons such as:



▼ **Select material from list** — material is selected from a reference file of densities (...IASCOKOMPAS-3D V...Sys\Graphic.dns; file contents can be edited by the user),



▼ **Select material from reference** — material is selected from an external reference (the reference should be connected to the system KOMPAS-3D).



In the system KOMPAS-3D, there are ten properties of **Replacement for material** and ten properties of **Coating**. This means ten different replacements for material can be selected and up to ten types of coating can be assigned in one document.

▼ Author*, Organization*, Commentary*

Values of properties are related to the data displayed in the **Information about document** dialog (see Section 1.3.5 on p. 73) in the current document. They can be set manually and edited in the document.

▼ Object type

The system automatically determines the type to which the document or object belongs: *Drawing*, *View Insertion*, *Fragment Insertion*, *Fragment*, *Macro Element*, *Assembly Unit*, *Part*, *Standard Item*, *Library Component* or *Body*.

▼ Ref.No.

Model components and bodies can have their positions marked. Numbers of these positions are used as the *Position* property values for components and bodies.

In a graphical document, link between the position number and the object is ensured via the specification object. See Section 6.1.3 on p. 1459 for more details on BOM items in documents.

▼ **File Name, Full File Name**

Properties reflect the file name and full path to the document's or object's file. For objects, which have no file (for example, for bodies, graphical macro elements), path to the current document is saved in the property's value.

▼ **Created, Last Modification**

Creation and last file modification dates are filled in automatically.

▼ **BOM section***

The property that reflects the BOM section name for the object included in the document. After BOM object creation value is filled in automatically.

▼ **Designed by*, Checked*, Approved*, Technological control*, Compliance assessment***

Value is set manually. In a drawing, values can be set in the title block.

Values set in the model document are transferred to the properties of the drawing created on the basis of the model (see the *Model properties in the associated drawing* on p. 1388 section).

▼ **Accuracy class**

The value reflects the class of accuracy selected during creation of the model (see Section 9.2.7.4.9 on p. 1854).

In a graphic document, the value is set manually.

5.1.3.2. Additional properties

Advanced property is a property from the external property library. Advanced properties are created and assigned to a document by the user.

Values of complementary properties are also assigned by the user. The value assignment method depends on its type.

- ▼ The value of the *Integral* or *Real-valued* type is entered in the field as numbers (manually or using the spinner) or is chosen from the value list.
- ▼ The value of the *String* type is entered in the field as text or is chosen from the value list.
- ▼ The value of the *Logical* type is assigned using the option located in the *Value* cell of this property type. If the option is enabled, the property has the *Yes* value; if the option is disabled, the property has the *No* value.



The property value type and the data required to assign the value (value range, value list) are assigned when the property is created (see Section 5.1.2.5.1 on p. 1375).

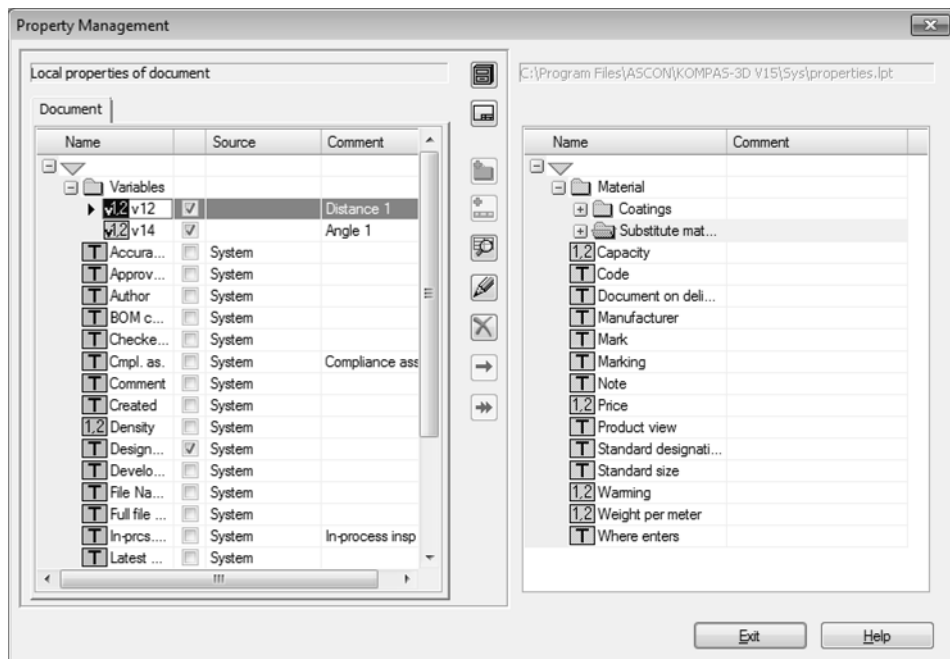
Values of additional properties may obtain values by a link to another object in the document. The links in properties are detailed in Section 5.1.4.4.1 on p. 1395.

5.1.3.2.1. Properties created on the basis of variables

In a model, a property can be created on the basis of a variable. The value of such property is changed automatically when the variable's value changes. And, vice versa, if you edit the property's value, the value of the variable will also be changed resulting in the respective rebuilding of the model.

You can create properties on the basis of all variables, except for functional and interval ones (see Section 7.1.5.1 on p. 1589).

In order to create and add a property from a variable to a document, select the required variables in the Variables Window and call the **Add to the list of properties** command from the context menu. A “checkmark” will appear next to the command name. It means that the prop-



erty based on this variables has been created in the document. Calling the command again removes the property.

The created property and its value appear in the document's list of properties. The property value's type is defined automatically. The following types are possible:

- ▼ *Real* — for a property created on the basis of any variable, except for the variable of the **Exclude from calculation** parameter,
- ▼ *Logical* — for a property created on the basis of a variable of the **Exclude from calculation** parameter,

You can view the list of properties created on the basis of variables in the **Property control** dialog of the **Document** tab (Fig. 5.1.10). The **Variables** section is automatically created for these properties in the dialog's list of properties. All properties created on the basis of variables are placed in that section. Names of properties and commentaries to them correspond to the name of variables and their commentaries in the Variables Window.



Properties included in the **Variables** section of the **Property Control** dialog are available for removal only. These properties cannot be edited, copied and moved.

When deleting a property, the variable, on which basis the property is created, will not be deleted.

When deleting a variable, only the property value, created on the basis of the variable, will be deleted. The property itself remains in the document's list of properties. Properties should be deleted manually in the **Property control** dialog or using the **Add to property list** command (see above).

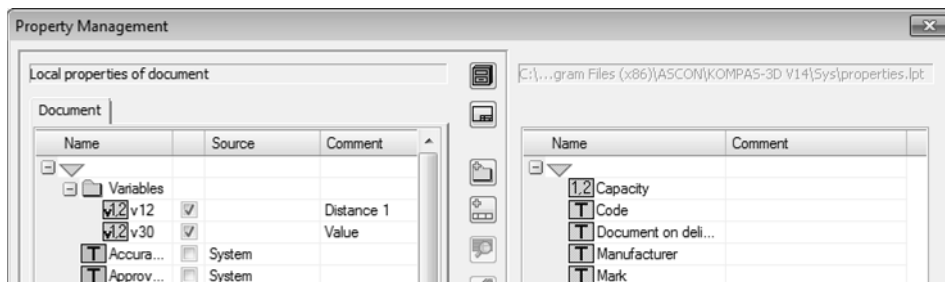


Fig. 5.1.10. Properties on the basis of variables in the **Property control** dialog

In order to change the values of properties created from variables, the same methods are used as for changing the value of other document properties (for details, refer to Section 5.1.4.4.5 on p. 1399).

5.1.4. Work with Properties

You can set and view properties and their values to a document or its objects as follows:

- ▼ using controls of the Property bar — this method makes it possible to work with properties of an individual object or document (see Section 5.1.4.1 on p. 1385);
- ▼ in the Property Editor — this method makes it possible to work with properties of all objects of the document, including the document itself (see Section 5.1.4.2 on p. 1389).

Object properties are saved in the current document without being transferred to the files of objects. By default, model components and insertions in a graphic document are displayed with the properties of the source that can be changed, if necessary. Changed values are not transferred to the source files.

If an object is used in multiple documents it may be rational to specify properties in a source file. In this case all the changes made in the object file are transferred to the documents.



It is recommended to use the property libraries when working with properties.

Assigning property values is detailed in Section 5.1.4 on p. 1384.

5.1.4.1. Specifying values of properties in the Property bar

5.1.4.1.1. General Working Techniques



After calling the **Properties of a model** command (**Properties of a drawing**, **Properties of view insertion**, etc.), the Property bar displays property controls including the **Property list** (the bar is located on the **Properties** tab).

The bar contains a list of shown properties of the current document (see Section 5.1.2.5.6 on p. 13805.1.11) and the list controls (Fig.

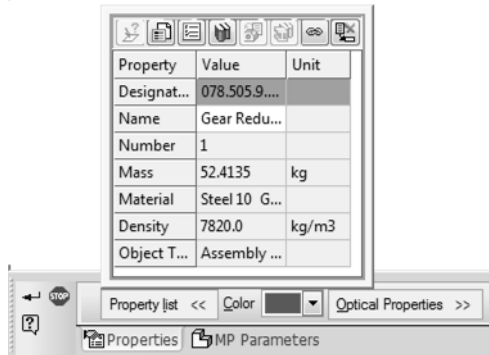


Fig. 5.1.11. Viewing Assembly Properties

The document property list is displayed as the table containing property names, their values and units of measurement.

Cells of properties that are not available for editing in the **Property list** bar have gray background.

When you view properties of an object included in the document, cells of the *Source* column of some properties display options (Fig. 5.1.12).). This means that you can obtain a property value from the source— object file. An option can be displayed as follows.

- ▼ The option is enabled. The property value is obtained from the source and can be replaced with the value assigned in the current document. After the value is replaced, the option is disabled automatically.
- ▼ The option is disabled. The property value is assigned in the current document on the **Property List** bar. Enable the option to obtain a property value from the source.
- ▼ The option is enabled and inaccessible. The property value is obtained from the source. You cannot modify the property value on the **Property List** bar.
- ▼ The option is disabled and inaccessible. The property value is assigned in the current document (e.g. in the model the component mass value is assigned on the **MP Parameters** tab of the Property Bar). You cannot modify the property value on the **Property List** bar.



When needed, you can view all source properties by clicking the **Source Properties** button. The button is located on panel in a model only. Upon clicking it, the information window containing the current date, full component file name and its property list appears.

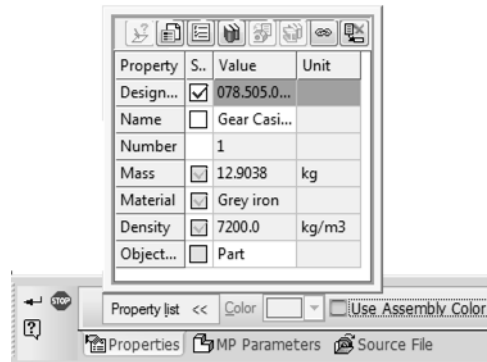


Fig. 5.1.12. Viewing Model Component Properties

The property list includes both system and complementary document properties.

Assignment of system property values is described in Section 5.1.3.1 on p. 1380, assignment of complementary property values is described in Section 5.1.3.2 on p. 1382.



Pay attention to the following specifics when assigning property values of objects having source.

- ▼ The default property value is the value transferred from the source— component file, drawing or fragment.
- ▼ Values of properties set in the document for its objects are not transferred to source files.



The property values of the current document assigned by the user on the **Property List** bar can be deleted. To delete a value, select it in the property list and press the **Delete Value** button.

If a value can be obtained from a source, the property value field will automatically get a value following deletion of the previous one.



To create a link in a property's value to an object in the document, highlight the property in the list and click the **Insert link** button. For details on links in properties, see Section 5.1.4.4.1 on p. 1395.

You can edit the property list of the current document and customize display of properties in the list without exiting the **(Document) Properties** command.



- ▼ The property list is edited in the **Property Management** dialog (see Section 5.1.2.3 on p. 1372). You can call this dialog by clicking the button of the same name.



- ▼ Customization of displaying of properties can be done in the **Setting Properties List** dialog (see Section 5.1.2.2 on p. 1372). You can call this dialog by clicking the button with the similar name.



To view all values of the document's objects' properties, use the Property Editor (see Section 5.1.4.2 on p. 1389).

5.1.4.1.2. Work in a model document

During work with the model document, property values can be set both for the model itself and for its bodies and components.



To set attributes to a model documents, use the **Model attributes** command. The command can be called from the **File** menu or from the context menu of the root object of the Model Tree.

Setting model object properties is possible via the **Component Property** or **Body Properties** command. The command can be called from the context menu of the object selected in model window or in the Model Tree.

After calling the command, the Properties panel displays control elements that allow to assign property values of a model document or of an object, included in it.



To transfer properties of components to the assembly document, copy their properties in the **Property management** dialog on the **Components** tab (see Section 5.1.2.5.5 on p. 1379).



In order to set properties for versions, use the **Version number** command. This command is called in the Tree of Versions, the version's context menu. Working with the **Version properties** command is the same as working with the **Model properties** command.

For more information about version property editing, refer to Section 2.13.3.4 on p. 763.

General methods of setting properties are described in Section 5.1.4.1.1 on p. 1385.

5.1.4.1.3. Working in a drawing

You can set properties in a drawing to the document and its view insertions, fragment insertions and macro elements.



Drawing Properties command allows viewing and editing the properties of a drawing document. The command can be called from the **File** menu or from the context menu of the root object of the Drawing Tree.

Using **Macro element's properties**, **View insertion properties**, **Fragment insertion properties** commands, you can set the properties for the document's objects. The commands are called from the context menu of the object selected in the Drawing Tree or in the drawing window.



To transfer properties of view insertions and fragments to the drawing document, copy their properties in the **Property management** dialog on the **Insertions** tab (see Section 5.1.2.5.5 on p. 1379).

System property values are defined automatically. *Position* and *Specification section* properties are filled out, if a specification object is created which is linked to a macro element or an insertion.



Values of such system properties as *Designation*, *Name*, *Material*, *Mass*, *Designed by*, *Checked*, *Approved*, *Technological control*, *Compliance assessment* can also be specified in the title block. Once it has been filled out, values of these properties are displayed in the list of the document properties. After editing values using the **Drawing properties** command in the Property Editor, all changes will be transferred to the respective cells of the main text.

General methods of setting properties are described in Section 5.1.4.1.1 on p. 1385.

Model properties in the associated drawing

The following properties are transferred to the associative drawing from the document model. These properties can be conditionally divided into groups that differ from each other by the principle of their handling.

▼ *Designation*, *Name* properties.

Values of properties specified in the model document are transferred to the model's drawing, including the title block cells, and, vice versa, values from the drawing document or from its title block are transferred to the model's properties.

Values can be modified anywhere: in the drawing, using the **Drawing's properties** or **Title block** commands, in the model, using the **Model properties** command.



See Section 3.6.3.8 on p. 1231 for more information about filling the title block of an associative drawing.

▼ *Mass*, *Material*, *Density* properties.

Values of properties are set in the **MCC parameters** tab of the model document and transferred to the model's drawing, including the cells of the title block.

The values can be edited: in the drawing, using the **Drawing's properties** or **Title block** commands, in the model, using the **Model properties** command.



Together with the *Material* property, its hatching parameters are also transferred to the drawing — hatch style, color, and angle that are set in the model for each material (see Section 3.6.2.7.1 on p. 1220).

Values of properties are not transferred from the drawing to the model.

▼ *Designed by**, *Checked**, *Approved**, *Technological control**, *Compliance assessment** properties.

Values of properties that are set in the model document are transferred once in the process of creation of a drawing to its properties and title block (as long as they had not been entered in the drawing previously). During further editing, no transfer of property values from the drawing to the model (or from the model to the drawing) takes place. Values have to be edited manually, for example, using the **Drawing's properties** command in case of the drawing.

▼ Properties of components and bodies of the model which is the basis for a drawing.

Available as read only. In order to view the list of properties, **Component properties** and **Body properties** commands are used. They can be called from the model context menu.

Properties included for display in the drawing document are mentioned in the list. To change the list of displayed attributes, use the **Property management** command (see Section 5.1.2.3 on p. 1372).

The Properties of a Graphic Macro Elements

You can set attributes to graphical macro elements in a document or in the fragment library.



For this use command **Macro element Properties**. The command can be called from the context menu of the macro element.

The properties set for the macro element will be transferred if it is copied or inserted.

5.1.4.1.4. Work in the fragment

You can set properties in a drawing to the document and fragment insertions and macro elements.



Fragment Properties command allows viewing and editing the properties of a fragment document. The command can be called from the **File** menu.

Using **Fragment insertion properties** and **Macro element properties**, you can set the properties for the document's objects. The commands are called from the context menu of the object selected in the document window.



To transfer properties of fragment insertions to the fragment document, copy their properties in the **Property management** dialog on the **Insertions** tab (see Section 5.1.2.5.5 on p. 1379).

General methods of setting properties are described in Section 5.1.4.1.1 on p. 1385.

5.1.4.2. Property Editor

The Property editor makes it possible to:

- ▼ view and edit the object properties in the current document.
- ▼ create and add new properties;
- ▼ save property values into a separate file or print them out.



The Property Editor appears on the screen after calling the **Property Editor** command which can be called when working with model documents or graphical documents by clicking the button on the **Reports** bar or by choosing the name from the **Service — Reports** menu.

Upon calling the command, the **Property editor** dialog will appear (Fig. 5.1.13).

There is a window in the dialog, where objects and their properties are displayed in the form of a table. Control elements of the dialog make it possible control display of attributes in the Editor and perform different operations to them.



Fig. 5.1.13. Editor of properties in model dialog

5.1.4.2.1. Object Display

In the *Name* column there is the list of objects, that have a Tree structure.



If the **Group** button is clicked, objects are displayed as a group. This means that objects of the same level with the same properties are represented by one line. Number of group objects is summed and displayed in the *Number* property. As other properties' values, values of one object of the group are displayed. The rows with non-matching with other property remain unchanged. Icons for grouped objects are listed in Table 5.1.3.

If the button is released, then the full list of objects is displayed.

In the drawing, if object grouping is enabled, grouping of objects by views is possible. For this purpose, click the triangle on the right of the grouping button and call the **Group by views** command in the appearing menu. Once the command is called, all objects are displayed grouped by their views. Names of views are available for editing.

If several views of a single model are present in a drawing, the model is displayed in several views of the document. In the *Number* property of such model, the value of one will always be displayed.

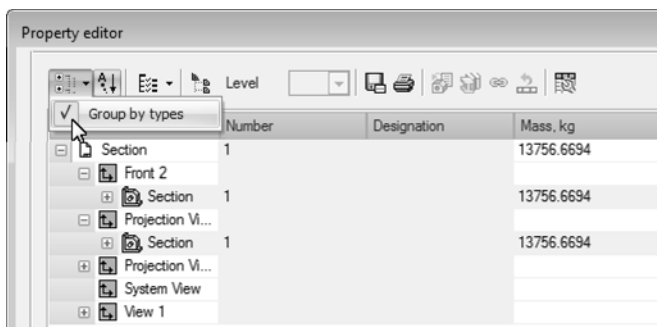


Fig. 5.1.14. Calling the command for grouping by views in a drawing



Use the **Sorting** button to control object sorting. Objects in the dialog are sorted by their types whether the button is clicked or not.

If the **Sorting** button in pressed, then the each type objects are alphabetized. If the button is released, then the objects are displayed in the order of their creation.



In order to enable or disable displaying objects of different types in the Editor, click the **Design Tree Contents** button.

A menu containing the list of objects' types will appear on screen. A tick on the left of a name means that displaying of objects of the respective type is enabled; if there is no tick, it is disabled.

If the **Display all** variant in the menu is enabled, all types will be displayed.



To set the level to which (inclusive) objects making up assembly items of models are displayed, click the **Level** button. The list to select level number, becomes available.

By default, the button is not pressed. This means, that objects from all model levels are available for editing.

The **Level** element is located in the Editor when working in a drawing or assembly.

Tab. 5.1.3. Icons of grouped objects in the Property Editor

Icon	Types of grouped objects
	Parts
	Subassemblies
	Library components
	Standard products
	Local parts
	Insertions of views in a drawing
	Insertions of fragments in a drawing
	Macro elements in a drawing

5.1.4.2.2. Column Setting

By default, there are columns with properties connected to the document in the Property Editor. Each column in the Editor corresponds to one property.



In order to modify the set of properties displayed in the Editor, click the **Customize columns** button. The dialog will appear where you can create a set of displayed properties and adjust their mutual position (Fig. 5.1.15) .

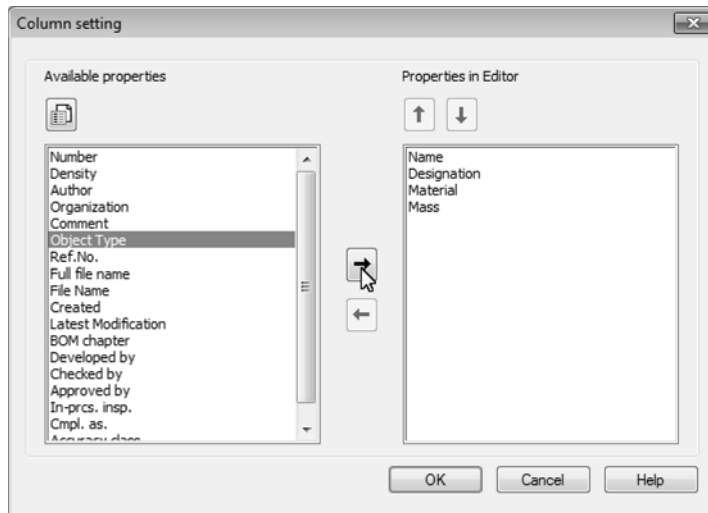


Fig. 5.1.15. Column setup dialog in the Property Editor

The dialog lists the local document properties. As part of them, in the **Available properties** window are the document's properties, available for selection, while in the **Displayed Properties** are properties selected for display in the Editor.



To add a property to the list of displayed properties, highlight it in the **Available properties** window and click the **Move to the right** button. The property will be moved to the **Displayed properties** window.



To remove a property from the list of displayed properties, highlight it in the **Available properties** window and click the **Move to the left** button. The property will be deleted from the window, and it will appear in the **Available properties** window.



If necessary, you can change the list of available properties. For this purpose, click the **Property management** button. The **Property management** dialog will appear on the screen making it possible to add a property to the document, create a new property, attach a new or previously created property. Attached property appears in the **Displayed properties** window. For details on working in the **Properties Management dialog** see Section 5.1.2.3 on p. 1372.



You can change the order of properties in the Editor by moving them in the **Properties in the Editor** window. To do this, select the property, the position of which you need to change, and click **Move Up/Down** button. The button is available, if it is possible to move the property to the specified position.



The *Name* property can not be moved or deleted from the list of displayed properties.

Upon finishing columns setting up, click the **OK** button.

5.1.4.2.3. Select cells and rows

Highlighting cells may be required for setting property values, including insertion of links, copying, deleting and other actions with respect to the cells' contents. Contents of highlighted

cells or lines can be inserted into other documents of Windows applications through the exchange buffer.

To select a cell, click it with the mouse. To select a row, click the mouse in the field on the beginning of the row (see Fig.5.1.13).

To select several cells or rows, click the mouse on them holding the <Ctrl> key. To select several cells or rows at once, click the first and the last cell or row being selected while holding the <Shift> key.

Highlighting cells in a table is accompanied by highlighting the respective object in the document.

In order to deselect a cell, click any place in the window outside the table.

5.1.4.2.4. Editing of Values

You can edit property values of the document and its objects in the Property Editor. Unavailable for editing cells have a gray background¹. Property values from a source file are displayed in blue, specified in the current document are black.

- ▼ To change a property value click the mouse on the selected cell and edit the text in it. When editing grouped objects' properties, changes are transferred to all objects of the group.
- ▼ To return the property value from the source, select the cell and all the **Source** command from the context menu.
- ▼ To copy, paste, cut, delete the contents of the selected rows or cells, use the context menu commands. To insert a previously copied content of cells, highlight the same range of cells and call the insertion command.
- ▼ You can paste contents of separate cells or rows to another Windows applications by using the clipboard.
- ▼ Instead of a property value, you can insert a link to an object in the document. For more about link insertion and editing, see Section 5.1.4.4.1 on p. 1395.

You can modify material and its density, as well as the mass of objects. Detailed information about MCC editing in the Property editor can be found in Section 5.1.4.4.2 on p. 1395.

You can edit system properties of the document: *Author*, *Organization*, *Commentary*.



To cancel the cell editing, select the cell and press the **Source Text** button in the dialog. When you press the button, the cell is restored to the form that it had after dialog opening.

It is impossible to cancel editing, if the link was destroyed or if values in property cells are set in dialogs that are called from the Editor, for example: *Designation*, *Material*, *Mass* and others.

To transfer changes into the document and to update links, click the **Apply** button of the dialog. In this case canceling the changes is not possible.

5.1.4.2.5. Save Table to File and Print



To save the table to file, press the **Save As...** dialog button and choose file type.

- ▼ KOMPAS-Drawings (the *.*cdw* file),

1. The exception are model's properties Mass, Material, Density, Replacement for material, Coating, BOM section. Their cells have gray background, but their values can be edited.

- ▼ KOMPAS-Fragments (the *.frw file),
- ▼ KOMPAS-Text Documents (the *.kdw file),
- ▼ KOMPAS-Tables (the *.tbl file),
- ▼ Text Files (the *.txt file),
- ▼ ODF Spreadsheets (the *.ods file),
- ▼ Microsoft Excel Spreadsheets (the *.xls file).

When saving into a graphical or text KOMPAS document, text parameters preset when setting the report's table for the respective documents' type are used. For example, to set text parameters when saving table in a graphic document, call the **Service — Parameters... — New Documents — Graphic Documents — Report Table Parameters — Header/Cell** command. For Controls of the Dialog, see Section 9.2.4.3 on p. 1752.



To print the Editor's window contents, click the **Print** button. Set the print parameters in the dialog appeared on the screen and then click button **OK**.

5.1.4.3. Copying Properties

When copying objects, copied objects automatically obtain properties repeating properties of the source object.

For **Graphic objects**

- ▼ when dragging the object with the mouse and holding the <Ctrl> key— properties are always copied.
- ▼ when inserting a graphical object from the exchange buffer or when executing symmetry, scaling, shift, rotation commands, as well as copying by specification, by curve, by grid and so on—you can control the properties' copying.

Those system properties will be copied whose values are set by the user, as well as all additional properties. Assigning property values is detailed in Section 5.1.3 on p. 1380.

To control copying, use the **Copy Properties** group of switches on the **Attributes** tab of the Command Properties bar.



Object is copying without properties when the **Don't Copy** switch is activated.



If it is required to create a copy with the same properties as the properties of the source object, activate the **Copy** switch.



If a command provides for deleting or retaining source object, properties are copied if the **Retain source objects** option is selected in the **Mode** group of the Property bar.



If none of the objects selected for copying has user-defined properties, the **Copy Properties** switches will be inaccessible.

For **models**, properties are copied during objects copying, for example, when “dragging” the component's icon from the Tree to the Model window while the <Ctrl> key is pressed.

Also, when creating arrays, source object's (component's or body's) properties are copied into each array's instance.

5.1.4.4. Peculiarities of setting properties

5.1.4.4.1. Link in a property value

Instead the property's value, you can insert a link to the object within the current document that contains text or numbers. Inserted link can be edited or exploded.

You can work with links:

- ▼ in the Property bar when setting values of document or object properties (more information about setting values in the Property bar can be found in Section 5.1.4.1 on p. 1385);
- ▼ in the Property Editor (see Section 5.1.4.2 on p. 1389 for more information about the Property Editor).



To insert a link, select one cell with the property value and call the **Insert Link** command. The command may be called by clicking the button above the list or selecting it in the contextual menu of the highlighted cell in the Property Editor.

After calling the command, a dialog appears, in which you need to set up the link parameters. Setup is the same as when inserting a link into text (see Section 4.1.4.3 on p. 1337).

Once inserted, the property's value is replaced by the value obtained from the source of the link. Property value with a link is displayed in blue.

Values by link can obtain properties with any type of data, except for *logical* data.



It is recommended to insert link to data whose type corresponds to the type of data of the property's value. For example, if a property value's data type is *integer*, you should insert in this cell the link to an integer value in the source.

To edit the link, select the cell and call the **Edit Link** command from the context menu. The screen will display the same dialog as during creation of link. Introduce changes and close the dialog

To delete the link to the link's source, highlight the cell with the link and call the **Explode link** command from the contextual menu. As a result of exploding, the contents of the link becomes a property value that can be edited manually.

Link Explosion can not be undone.

5.1.4.4.2. Specifying a material in the model document. MCC Recalculation

You can edit values of the following system properties: *Material*, *Density*, *Mass*, as well as recalculate the ¹model MP.

You can do this in one of the following ways:

- ▼ using the **Model properties** command in the **MCC parameters** tab of the Property bar when specifying values of properties:
- ▼ in the Property Editor (see Section 5.1.4.2 on p. 1389 for more information about the Property Editor).

1. MCC recalculation means determining the model's mass taking account of changes of the current density, as well as the mass of components and bodies included in the model.

In the Property Editor, the same methods are used as in the **Model properties** command (see Sections 2.14.6.1-2.14.6.4).

In order to change material, density or mass of an object in the Property Editor, perform the following actions.

1. Call the object property editing dialog. To do this, complete the following steps:
 - ▼ highlight the cell with the value of the *Material*, *Density* property or *Mass* and click the **MCC parameters** button in the Editor's bar or
 - ▼ double-click the cell or
 - ▼ click the ellipsis button in the right section of the highlighted cell (Fig. 5.1.16).

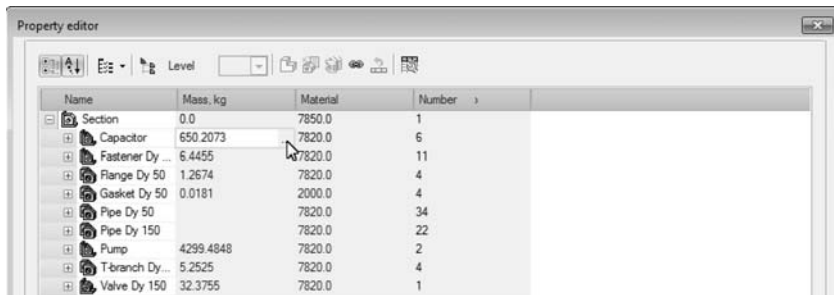


Fig. 5.1.16. Calling the material editing dialog in a model

A dialog will be displayed, in which you can make modifications of property values.

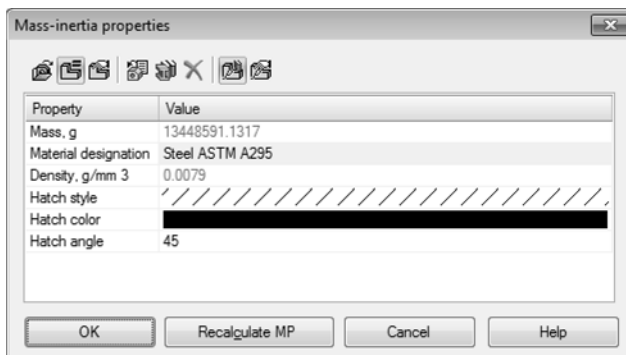


Fig. 5.1.17. Dialog for changing property values of MCC

The dialog contains the same controls as when setting the values of properties in the Property bar (in the **Model properties** command of the **MCC parameters** tab).

2. Choose the recalculation method for the object's MCC.
 - ▼ **From source** — enabling the switch means that value of properties for the component are imported from the source file; in case of a body, they are imported from the current document.
 - ▼ **Calculation by Density** — enable this switch to define a material and/or density of the material to calculate the model mass.





- ▼ **Calculation by mass** — enable this switch to specify the mass value manually. It is available in the dialog when setting up a part, assembly or assembly component.



When opening the dialog, the enabled switch defines the method applied in the current object's MCC calculation.

3. Specify new parameters: material, density or mass, as well as hatch parameters, if required.
In order to update the value of the object's mass, click the **Recalculate MCC** button in the dialog.
Once the button is clicked, MCC of the object (part, component or body) takes place taking account MCC modifications made earlier with respect to bodies and components contained in the object. The mass recalculation result is displayed in the **Mass** dialog.
Click the **Exit** button in the dialog.
4. Apply new properties to the model. For this purpose, click the **Apply** button in the Property Editor. Modified properties are displayed in the Editor.
5. Edit MCC of all the required objects as it is described in pp 1-4.
In order to recalculate MCC of the whole model in the Property Editor, call the dialog for its root object and click the **Recalculate MCC** button in it.



When setting the values of properties, the following should be taken into account.

- ▼ If a body is being setup, the specified material is considered to be the this body's own material.
- ▼ If a part is being setup, the current material will be applied to those of its bodies, which do not have their own specified material.
- ▼ If an assembly is being setup, the selected material will be applied only to the bodies created in this assembly (not to the components!), and only to those of them which do not have their own specified material.
- ▼ If an assembly component is being setup, then it makes sense to select a material only if this component is a part. The defined material is applied only to those of its bodies which have no own material specified.

Material properties, such as density and hatch parameters, including those manually set, are saved together with the material in the model and are applied to bodies of parts, components, assemblies in accordance with the rules described above.

In order to cancel editing and set values of object properties from the source, call the dialog for this object and enable the **From source** switch. Then, apply this change to the model by clicking **Apply** in the Property Editor.

5.1.4.4.3. Specifying a material in a graphical document

You can set *Material* and *Density* properties for graphic objects by choosing their values from the reference file of densities or from the external Reference connected to the KOMPAS-3D system.



The Reference file of densities *Graphic.dns* is supplied together with the system and can be found in the \Sys subfolder of the main KOMPAS-3D folder. The user can add and edit values in the file.

You can do this in one of the following ways:

- ▼ using the **Drawing properties** command (**Fragment properties**, **Macro element properties**, and so on) in the Property bar (see Section 5.1.4.1 on p. 1385 for more information about setting values of properties in the Property bar).
- ▼ in the Property Editor (see Section 5.1.4.2 on p. 1389 for more information about the Property Editor).



In order to set a property, select the cell with the *Material* property value and click **Select material from the list** or **Select material from reference** in the toolbar.



In the Property Editor, you can also select material from the reference by clicking the ellipsis button in the right section of the selected *Material* or *Density* cell or by double-clicking the cell.

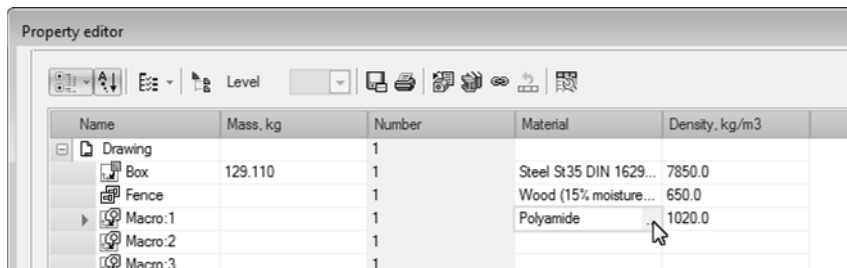


Fig. 5.1.18. Calling the material editing dialog in a graphical document

5.1.4.4.4. Specifying a designation

Document (object) designation can be **simple** or **composite**. A simple designation includes only the basic designation. Apart from the basic part, a composite designation includes the version number (when working with graphic documents and components — the version's additional number as well; when working with drawings — the code as well).

A simple designation can be entered or edited directly in the cell assigned to it, while a composite designation is handled in a special dialog, **Designation**, where there is a separate field for each part of the designation. Working with the dialog is described in Section 2.13.2.1.1 on p. 756.

You can specify a designation of a document or selected object:

- ▼ in the Property bar, when setting values of document or object properties (more information about setting values in the Property bar can be found in Section 5.1.4.1 on p. 1385); A simple designation is entered directly into the designation cell. To call the **Designation** dialog, select the cell and call the **Edit designation** command from the context menu.
- ▼ in the Property Editor (see Section 5.1.4.2 on p. 1389 for more information about the Property Editor). A simple designation is entered directly into the designation cell. To call the **Designation** dialog, select the cell and press the ellipsis button in its right part.

If the cell contains a composite designation, you can double-click the designation to call the **Designation** dialog.



Parts of a composite designation should be presented in different files and, respectively, the **Designation** dialog should be used if you plan to work with BOM and reports because such presentation provides adequate sorting. Otherwise, all parts of a composite designation can be entered as one line (which will be treated as a basic designation).

A composite designation “turns” into a simple one if fields with the version number, the additional number and the code (delimiters should be of system type or should not be present) are cleared in the dialog.

5.1.4.4.5. Specifying values of properties created on the basis of variables

The model's property created from a variable, receives the value of the variable. See Section 5.1.3.2.1 on p. 1383 for more information about creating properties from variables.

You can change the property's value using the methods designed for the document properties. For this purpose, controls of the Property bar (see Section 5.1.4.1 on p. 1385) or Property Editor (see Section 5.1.4.2 on p. 1389) are used.

Note that the property's value can be modified only if the value of the variable, from which the property is created, is set. If the value is defined by a constant, expression or link to the external file's variable, it is not available for editing. Also, it is impossible to edit properties created from informational variables and variables to which the **Read-only** constraint is applied.



Value of the property created from the variables of the **Exclude from calculation** parameter is modified by enabling/disabling the option (see Section 7.1.4.1.1 on p. 1587 for information about variables of objects' exclusion from calculation).

If a variable of an operation's parameter is used for creation of a property, and its value is expressed in linear or angular values, or the variable of the sketch's controlling dimension, you can assign/change tolerance for the respective value of the parameter or tolerance for dimension when working with the property in the Property bar or in the Property Editor.

The **Value and tolerance** dialog is used for assigning/changing tolerance. The dialog is called by the **Assign tolerance** command from the context menu of the cell with the property's value which is located in the Property bar or in the Property Editor.

The **Value and tolerance** dialog is illustrated in Fig. 2.13.6 on p. 769. Elements for assigning tolerance are described in Table 3.3.5 on p. 982; description of the **Value** field can be found in Section 2.13.3.5.4 on p. 768.

When edited, values of the property will be automatically replaced by the value of the respective variable and the model will be rebuilt as a result.

5.1.4.4.6. Setting BOM position and section

In a document, you can assign the *BOM section* property to objects which means that the object belongs to a certain BOM section. At the same time, BOM item is automatically created in the document (it is internal in case of an assembly).

The *BOM section* property can be specified using one of the following methods:

- ▼ in the Property bar (see Section 5.1.4.1.1 on p. 1385) — using the **Assign section** command which is called from the *BOM section* cell's context menu, or the *Position* command with the value of the property, or by double-clicking the cell;
- ▼ in the Property Editor (see Section 5.1.4.2 on p. 1389) — by clicking ellipsis button in the right section of the selected *BOM section* cell or the *Position* command with the value of the property, or by double-clicking the cell.

Once the command is called, the same dialog appears on the screen as when adding the base object to BOM (see Section 6.2.2.3.1 on p. 1472).

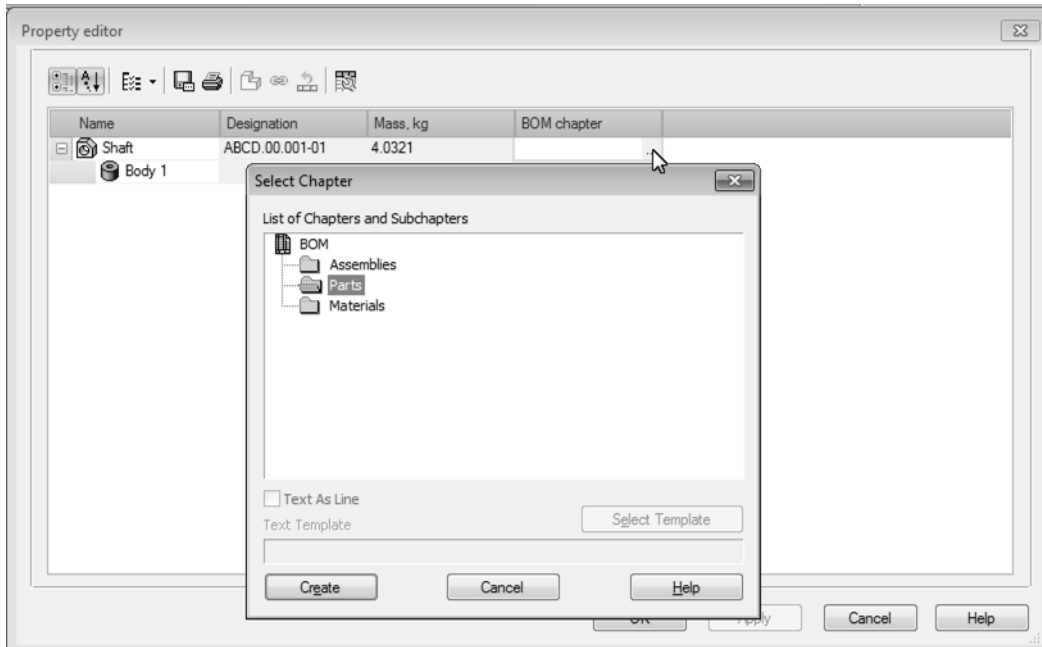


Fig. 5.1.19. Calling the dialog for selection of the object's BOM



In the dialog, choose the section and click **Create**. Then, confirm setting the property's values by clicking the **Create object** button in the Property bar.

For **Standard items** and **Materials** sections, you can fill the text part in the form of a line (default variant) or in accordance with a template.

You can choose the template in the dialog of section selection (Fig. 5.1.19). Calling the template is available when calling the command again, if the **Standard items** or **Materials** value of the *BOM section* property is set and the **Text part as line** option is disabled.

Once the template is selected, the **Template of text part filling** command becomes available in the *BOM section* or *Position* cell's context menu in the Property bar. The command makes it possible to set or edit text in the template's cells.

Upon calling the command, template editing dialog will appear. The dialog displays the table, in whose cells you can enter or edit the default values.

Set values in the table's field. If you need to use a default text in BOM, click any cell with a value. Confirm the text by clicking **OK**.



Fig. 5.1.20. Dialog for filling text part in by template

The text will automatically be included in the BOM item.

When setting *BOM section* properties, the following peculiarities should be taken into account.

- ▼ In a model, a property can be set for the whole model, component or body of an assembly; in a drawing, it can be set for view and fragment insertions, macro elements.
- ▼ You can edit the property's value, including replacement of the filling template, text editing. The section and name will be automatically changed in BOM.

After setting in the document, positions are displayed in properties of components or bodies in a model, or in properties of graphical objects in a drawing.

5.2. Reports

5.2.1. General information

KOMPAS-3D supports creating reports in the form of tables containing object data. Data about objects that are placed in the report are specified during creation of these objects as property values.

The reports can be used to issue various table documents without strict adherence to the standards of any one system or industry. When editing report data, object properties remain unchanged.

The data from the model-document or from the graphic document is transferred to the report table. For example, the report may contain the list of properties of parts – name, designation, mass, quantity, position, manufacturer, price, and so on. The list of parts and the set of properties to be placed in the report are defined by the user.

The objects used for report generation are components of any level (including standard and library items), bodies and graphic macro elements, view and fragment insertions.

The properties transferred to a report are system and complementary object properties. Property setting is detailed in Section 5.1 on p. 1369.

Report appearance is defined by the report style, with style parameters specified by the user.

The KOMPAS-3D system provides the following options to create reports:

- ▼ selection of objects in accordance with the specified criteria, such as type and property of the object, as well as limitation of the model's structure by a certain level (for associative drawings — by the model reading level);
- ▼ obtaining data on properties of various objects with summary calculation of their values, and also property editing;
- ▼ using the prepared report styles which can be edited during report generation, if necessary;
- ▼ report printing;
- ▼ placement of the obtained reports in the existing or newly created graphic or text documents of the KOMPAS-3D system and saving the KOMPAS Drawings (the *.*cdw* file), KOMPAS Fragments (the *.*frw* file), KOMPAS Text Documents (the *.*kdw* file), KOMPAS Tables (the *.*tbl* file), Text Files (the *.*txt* file), ODF Spreadsheets (the *.*od* file), Microsoft Excel Spreadsheets (the *.*xls* file) as independent files.
- ▼ creation of the associative report — a table linked to the document by whose objects it was obtained.

The buttons to call the commands to work with reports are placed on the **Reports** toolbar.



Fig. 5.2.1. The **Reports** toolbar

5.2.2. Preparation for creating a report

The report creation preparation includes preparation of the graphic objects and models to be reported, report style or style library creation, general setup of the KOMPAS-3D system.

- ▼ The object preparation involves pre-assignment of the required properties. Report properties are detailed in Section 5.2.2.1 on p. 1404.
- ▼ Report style customization includes setting of the parameters defining report appearance. Report styles can be created and saved to a library in advance. More about working with report styles in Section 5.2.2.2 on p. 1404.
- ▼ The KOMPAS-3D system setup consists in setting of the parameters required to place a report in documents. See detailed description of the system setup in Section 5.2.2.3 on p. 1416.



The report creation preparation is not required for the trial mode of work. You can use the default settings of the KOMPAS-3D system.

5.2.2.1. Report properties

The properties to be reported are local document properties. By default, these are system properties — *name, designation, material, mass, quantity, density, object type, file name, BOM section, date of creation* and others.

User can include additional properties from the libraries to the local properties list.

The default property library is *Properties.lpt*. The library contains additional properties— *mark, power, standard, price* and other properties. You can create several property libraries (*.lpt files) to generate reports of various purpose and add properties from them to a document.



Property values will be in the report if they are set before its creation. If the value is not set, the cell of this property will be empty in the report.

In reports that have a link to objects' properties (associative report, report with data in the form of links), properties' values can be also set after the report is created. They will be included in it automatically.

Property creation, list customization and property value assignment are detailed in Section 5.1 on p. 1369.



The **Properties Management** button of the **Reports** panel is used for calling the dialog in which you can edit the document property list. The dialog can also be called by the **Service—Libraries of Styles, Layout Types and Title Blocks— Object properties...** command.

5.2.2.2. Report Styles

5.2.2.2.1. General information

Report style defines report appearance. Report style settings affect the display of various properties in columns, column names, column sequence in a report, sorting and grouping methods, cell size and cell text alignment method, table pagination and other parameters.

You can create styles as follows.

- ▼ A style can be created or edited within the style library prior to report generation. For more information about style creating and customizing see Sections 5.2.2.2.2–5.2.2.2.7.
- ▼ The current style of the report can be set directly in the process of report creation. This style can be used only till the end of the document working session.

For multiple style use, save it to the library.

Report style libraries are the files external to documents (*.lrf). The libraries can be transferred to other workstations. Styles saved in libraries are available when reports are created in different documents.

It is required to connect library to the system to use the styles from it (see Section 5.2.2.2.8 on p. 1415).



In the KOMPAS-3D system there are **By Default** and **By All Properties** system report styles. These styles are available during report creation process (see Section 5.2.3.3 on p. 1418). They can be used to create a new style but cannot be disabled or deleted from the system or moved to any report style library.

5.2.2.2.2. Working with style libraries



Report Styles command allows to work with report style libraries. To call the command, press the button on the **Reports** panel or select its name from the **Service— Libraries of Styles, Layout Types and Title Blocks** menu.

After calling the command, the **Report Styles** dialog in which you can create, view, edit styles contained in the library and also edit the library structure, copy and move styles from one library to another appears on the screen.

Fig. 5.2.2 shows the report style library dialog.

The control elements of this dialog are represented in table 5.2.1.

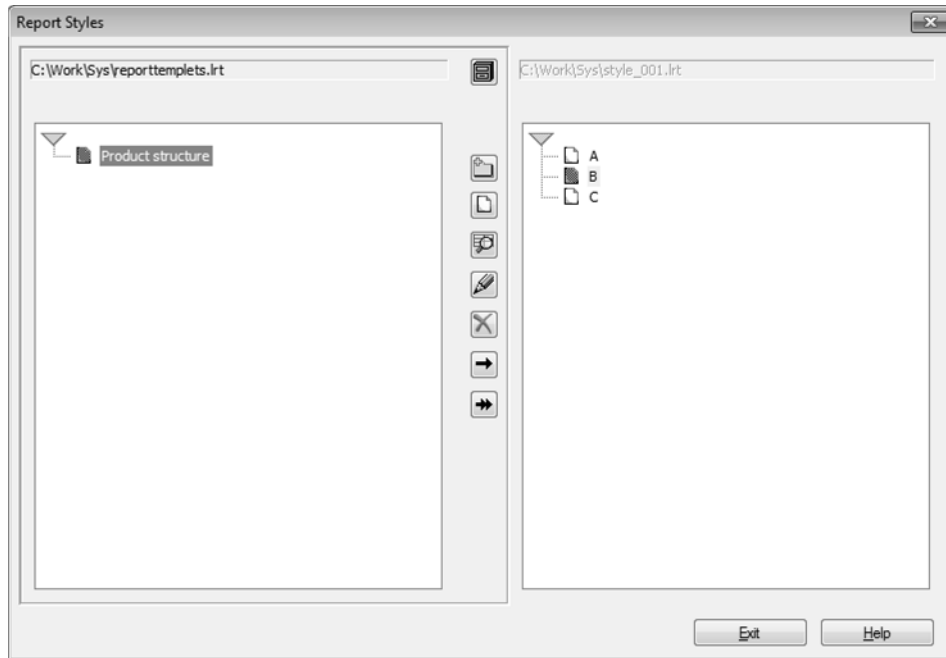












Fig. 5.2.2. Report style library dialog

Tab. 5.2.1. Report style library dialog

Element	Description
Preview Window	Serves to display a library style list. The dialog has two view windows, therefore it can display style lists of two different libraries at a time. Only one window can be active it is selected by a marquee. For example, the left window is active in Fig.5.2.2. To activate, click the mouse in the window. Style lists have a tree structure — style-containing sections, subsections, etc. The section nesting level is unlimited. All tree objects are automatically sorted by names within each section.
 Show Library	Allows to display, in the active window, the style list saved to the library and to create a new library. After pressing the button, you are offered to specify file name in the opened dialog.
 New Section	Allows to create a new subsection in the current section. After pressing the button, you are offered to enter the name of a new section (subsection) in the opened dialog.

Tab. 5.2.1. Report style library dialog

	Element	Description
	Create Style	Allows to create a new style in the current library section. After pressing the button, you are offered to perform new style-creation actions (see Section 5.2.2.2.3 on p. 1408).
	View	Allows to view a selected style. After you press the button (or double click the mouse) the style parameters setup dialog will pop up in the view mode.
	Edit	Allows to edit a selected object — (sub)section name or style. Upon pressing the button the screen will display: <ul style="list-style-type: none"> ▼ if a section was selected — the section name creation and editing dialog; ▼ if a style was selected — the style creation and editing dialog;
	Delete	Allows to delete a selected style or section. The section is deleted with all of its styles. Warning! A deleted style or library section cannot be restored.
	Copy* left/right	Allows to exchange styles or sections between libraries. After you press the button, the selected style or section will be copied to the adjacent window to the section which is current in it.
		
	Move* left/right	Allows to move styles or sections from library to another. After pressing the button, the selected style or section is moved to the current section of the adjacent window.
		

* The buttons are available if a library is opened in the inactive window.

To create a new library, do the following.



1. Press the **Show Library** button in the library dialog (Fig. 5.2.2). Enter a non-existing name in the appeared file selection dialog, press **Open** button and confirm the file creation. The active window of the style library dialog will be cleared since the newly created library is empty.



If you plan to store a large number of styles in the library, it is recommended to divide them into topical groups — sections; it will simplify future searches for the necessary style.

2. To create a section, press **New Section** button and enter section name in the appeared dialog. Generate the library structure by adding new sections.
3. To create style, select the section in the window that must contain the style being created. Click the **Create Style** button. In the opened dialog perform the actions to set parameters of the new style (see Section 5.2.2.2.3).

To finish the operation with styles, use the **Exit** button. After clicking it, all additions, deletions or changes will be automatically saved.

5.2.2.2.3. Customizing style

Report style customizing is done in the **Report Style Parameters** dialog. In the dialog you can customize and save the current report style to a style library.

The dialog appears on the screen

- ▼ upon pressing the **Create Style (View, Edit)** button in the library dialog (see Section 5.2.2.2.2 on p. 1405);
- ▼ upon pressing the **Customize...** button in the Property Bar of the **Create Report** command including the button in the Data Processing Window.

Fig.5.2.3 shows the report style parameter setting dialog.

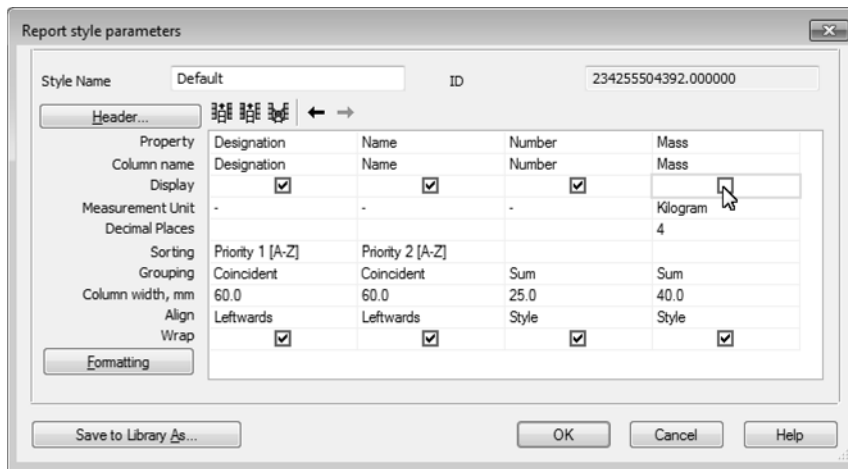







Fig. 5.2.3. Report style parameter setting dialog

The control elements of this dialog are represented in table 5.2.2.

Tab. 5.2.2. Report style parameter setting dialog

Element	Description
Style name	The field displaying style name. Enter the style name to save style to the library.
ID	The reference field containing a unique number automatically assigned by the system to the style being created. This number will be displayed in the dialog when the created style is further viewed or edited.
Header...	The button to call the dialog for setting up the table header. On customization, see Section 5.2.2.2.4 on p. 1411.

Tab. 5.2.2. Report style parameter setting dialog

Element	Description
Setup window	<p>The window displaying style settings as a table. The table contains</p> <ul style="list-style-type: none"> ▼ columns by the number of object properties — a property name is in the first table row; ▼ rows with cells containing controls for setting each property — a setting name is to the left of the window, and the setting result is displayed in a cell. <p>Columns of the By Default style contain the <i>Designation</i>, <i>Name</i>, <i>Quantity</i>, <i>Mass</i> properties, and columns of the By All Properties style contain all local document properties.</p> <p>The window is empty when a new style is created. To fill the window, press the Insert Column to the Left/Right button required number of times and edit settings of each column. Window controls are described in Table 5.2.3 on p. 1410.</p>
 	<p>Insert Column to the Left/Right</p> <p>The button that serves to insert a column with default settings to the left/right of the current one.</p> <p>If the window is empty, a column with the <i>Designation</i> property will be displayed in it.</p>
	<p>Delete Column</p> <p>This button deletes the current column.</p>
 	<p>Move Column left/right</p> <p>This button moves the current column left/right and swaps places with the adjacent one. The button is available if the table has at least one column to the left/right of the current one.</p>
Save to Library as...	<p>The button to call the dialog to save a style to the style library. In the opened dialog specify the file of the existing library or enter name of a new library. The style will be saved with the name specified in the Style Name field.</p>

Tab. 5.2.3. Parameter Setting Window Controls

Element	Description
Property	<p>Object properties field. Select a property option from the list which contains all local document properties.</p> <p>If there is no desired property in the list, click the mouse in the Customize... row. The dialog allowing to customize the list and select a property from the properties library appears on the screen. Complete the following steps.</p> <ul style="list-style-type: none"> ▼ To add property from the library to the list, attach library on the Property Libraries tab. The work in the dialog is similar to the property list customization for new documents (see Section 9.2.6.5 on p. 1762). ▼ Enable the options next to the desired properties on the Properties tab to customize the list. The highlighted property automatically becomes the selected property option in the report column.
Column Name	<p>The column header row in a report. For instance, the <i>Mass</i> property can have the <i>Mass, kg</i> header in a report.</p> <p>By default, the column name coincides with the property. Enter column name.</p>
Show	<p>The option allows to show/hide a column in a report. If the option is enabled, the column is shown in a report. If the option is disabled, the column is hidden.</p>
Unit	<p>The property unit field. Select a measurement unit from the list.</p>
Decimal digits	<p>The field to specify number of decimal digits for real values.</p>
Sorting	<p>The field displays sorting parameters.</p> <ul style="list-style-type: none"> ▼ If the field is empty, you cannot sort by this property. ▼ If the field contains text, this property participates in sorting. For example, Priority1 "A-Z" means that the property has the main (1st) priority, and its cells are sorted either in the alphabetical order. The dialog in which you can customize data sorting is called by clicking within the field. On the sorting customization, see Section 5.2.2.2.6 on p. 1413.
Grouping	<p>The field of the parameter which is used to group objects in the report. The field displays the default grouping method. For instance, the Match method for names means that all objects with the same names are presented in one row. Select a method.</p> <p>Grouping methods are detailed in Section 5.2.2.2.5 on p. 1412.</p>
Column Width, mm	<p>The report column width field in millimeters. Please enter an integer.</p>

Tab. 5.2.3. Parameter Setting Window Controls

Element	Description
Alignment	The field* of alignment parameter in column cells except for the header cell. Select an alignment method. Parameters of the style method are set in the text parameters setting dialog of the document where the report is located (see Section 5.2.2.3.2 on p. 1416).
Wrap	The option allowing to set up display of cell text with length exceeding column width. If the option is enabled, spaced text fragments can be carried over to a new row. If the option is disabled, cell contents are displayed shrunk in one row.
Layout	The button to call the dialog for setting up data report layout parameters. On the layout customization, see Section 5.2.2.2.7 on p. 1414.

* The control allows to select an option from the drop-down list. After you click a cell, in its right corner there appears a button with a triangle. Press the button to display the list for selecting a parameter option.

Settings for the current style remain valid to the end of your working session with KOMPAS-3D.

5.2.2.2.4. Setup of the table header

In order to set up the table's header, the **Table header** dialog is used (Fig. 5.2.4). It is called by clicking the **Header...** button.

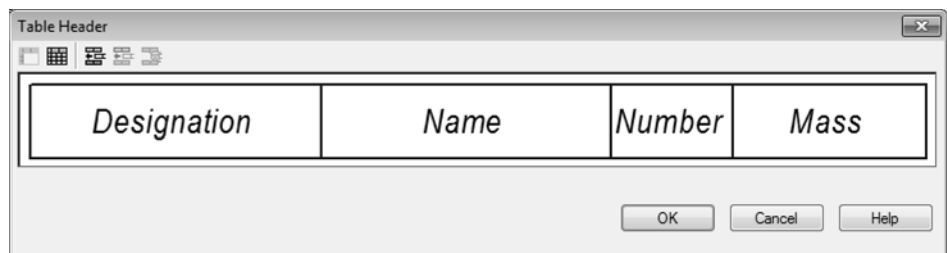


Fig. 5.2.4. Table header **dialog**

The dialog contains a field for the table header setup and a toolbar.

By default, the dialog's field displays only the headers of columns specified in the **Report style parameters** dialog.



To add a new row, click the **Insert a Row Above** button. An empty row will appear in the dialog's window. Later on, you can use this command for adding rows, as well as the **Insert Row Below** command. Adding is performed with respect to the row in which the cursor is positioned.

Add the required number of rows.



If you need to delete one of the added rows, position the cursor in it and click the **Delete row** button.

The number of cells in a row is the same as the number of columns in the table. You can merge and split cells.

In order to merge cells, highlight them. The cells are highlighted by the mouse as is described in Section 4.2.1.1.2 on p. 1350.



In order to merge highlighted cells, the **Merge cells** button is used. For more information about merging cells, see Section 4.2.1.1.3 on p. 1351.



In order to split a cell into several cells, position the cursor in it and click the **Split cells** button. The dialog will appear for adjusting the parameters of cell splitting. Working with the dialog is described in Section 4.2.1.1.4 on p. 1351.



Only the cells of added rows can be merged and split.



You can change the size of cells by “dragging” their borders using the cursor (see Section 4.2.1.1.7 on p. 1354). The style of the selected cells' borders can be set up in the **Borders** dialog, which appears on the screen after calling the **Borders...** command from the context menu. Working with the dialog is described in Section 4.2.1.1.8 on p. 1355.

Text in the table header's cells is entered in a normal way (see Section 4.2.1.1.1 on p. 1350). You can enter text in the cells of added rows and edit text of columns' headers which is specified in the **Report style parameters** dialog.

After customizing the table's header, close the **Table's header** dialog by clicking **OK**.

5.2.2.2.5. Data grouping methods

Grouping is combining of the objects with certain property values to display them in a report in a compact way. When grouping, several property lines are automatically replaced by a single line (hereinafter— *a resulting one*).

The following grouping methods are available.

Match — grouping of rows with the same values of the selected column only. At that, the property value is displayed in the *resulting* cell/cells of each group. The rows with non-matching with other property remain unchanged.

Sum — grouping of all rows of this column. At that, the *resulting* cell/cells of the group will contain the sum of the property values.

Range — grouping of all rows of this column. At that, the *resulting* cell/cells of the group will contain the minimum and maximum property values separated by ellipsis.



The **Sum** and **Range** methods are available for numerical properties only, e.g. *Quantity* or *Length*.

Enumeration — grouping of all rows of this column. At that, the *resulting* cell/cells of the group will contain one and all property values separated by a semicolon.

Note that grouping allows to combine objects by several properties at one time.

If the **Match** method is applied to any property, the **Sum**, **Range**, **Enumeration** methods automatically work within each group obtained by combining of similar designations.

For instance, you can set the **Match** grouping for designations and names, **Sum** — for quantity, **Enumeration** — for manufacturer. In the report rows, there will be groups of parts with both similar designations and names. In each group, the *Quantity* column will display the total number of parts and the *Manufacturer* column will list the factories.

5.2.2.2.6. Data Sorting

Data sorting serves to display report objects by groups formed from objects with similar properties. Here, the objects in each group can be arranged in alphabetical order (direct or reverse), either in ascending/descending order of their numerical values or according to the list of values.

The report data sorting is customized in the dialog that opens after clicking within the **Sorting** field in the **Report Style Parameters** dialog.

There are three sorting levels:

- ▼ **Priority 1 (Level 1)**— all object sorting.
- ▼ **Priority 2 (Level 2)**— sorting within resulting groups;
- ▼ **Priority 3 (Level 3)**— sorting within resulting subgroups.

The **By Default** style provide the first level sorting by designation, and the second level — by name in alphabetical order.

To change sorting setting, complete the following steps.

1. Customize the first level. To do this, expand the **Level 1** list. The list contains the object properties that are in the **Report Style Parameters** dialog window and the row with the **No** option to cancel sorting.

Select a property or cancel the sorting.

2. Select object arrangement option within a group.

A-Z — in alphabetical order or in ascending order of numbers,

Z-A — in reverse alphabetical order or in descending order of numbers.

By list — in the order displayed in the list of property values. The option is available if you select the property in the **Level1** list that has been created with the preset list of values in the **Property Parameters** dialog. Creation of properties is detailed in Section 5.1.2.5.1 on p. 1375.

3. Customize the second and the third sorting levels similar to the first one using the **Level 2** and **Level 3** elements.

The sorting customization result is displayed in the **Sorting** row of the **Report Style Parameters** dialog in the cells of properties involved to the sorting.



You can apply the grouping and sorting together to the same property in the report. The **Match** grouping method has to be selected.

5.2.2.2.7. Layout Setting

Layout customization in report style is performed in the dialog that appears on the screen after pressing the **Layout** button in the **Report style customization** dialog.

The control elements of this dialog are represented in table 5.2.4.

Tab. 5.2.4. Layout Parameters Setup Dialog

Element	Description
Build Bottom Up	The option allowing to control row placement order in a report. If the option is disabled, the header and rows are going top down. If the option is enabled, the header is placed on the bottom row and rows are going bottom-upwards.
Show Header	Group of elements allowing to control column header placement in a report. If Show Header option is turned on, on every table page the row with headers specified in the Column Name field of the Report Style Parameters dialog is placed. Row height can be set in the Table Header Row Height, mm field. If the option is disabled, the row is missing.
Table Header Row Height, mm*	The field to enter or select with the counter the header row height.
Row height of table cell, mm	The field to enter or select with the counter the row height.
Column Numbering	Group of elements allowing to control column numbering. If the Column Numbering option is disabled, there is no column with numbers. If the option is enabled, the first row with column numbers is automatically added to the table. To adjust numbering parameters, Number format and Begin with fields are used.
Number Format*	The list from which you can select a number format— letters or numbers.
Begin with*	The field to input or select (in the counter) start number in the column numbering.
Row Numbering	Group of elements allowing to control row numbering. If the Row Numbering option is disabled, there is no column with numbers. If the option is enabled, the first column with the <i>N</i> name and row numbers will be automatically added to the table. One-level and multilevel numbering is performed, depending on the chosen variant.
One-Level*	The numbering option that provides successive numbering of all report rows in ascending order.

Tab. 5.2.4. Layout Parameters Setup Dialog

Element	Description
Multi-Level*	The numbering option that provides independent row numbering within each level. It is applied for models that contain components. For example, first level assembly components will have numbers <i>1, 2, 3, ...</i> , and their respective integral components will be numbered <i>1.1, 1.2, 1.3, ..., 2.1, 2.2, 2.3, ..., 3.1, 3.2, 3.3</i> , etc.
Paginate	Group of elements allowing to control table pagination. If the Paginate option is enabled, the table is automatically split into several pages with equal number of rows in each page. Number of rows can be set in the Rows on Page field. You can also control the presence of empty rows on the last page. If the option is disabled, the report appears as one table.
Rows on Page*	The field to enter or select with the counter the number of table rows on one page.
Leave Blank Lines*	The option allowing to specify appearance of the last page. If the option is enabled, the last page can display rows that remained blank after the last object.

* Control element is available if the option in this group is enabled.

5.2.2.2.8. Connecting Report Styles

In order to enable styles from the library list, call the **Service — Parameters... — System — Reports — Setup of report style list** command.

The control elements of this dialog are described in Section 9.1.12.1 on p. 1742.

The general procedure for applying a style is as follows.

1. Add the required report style library file on the **Style Libraries** tab. To do so, click **Add**.
2. Enable the option located next to the added library name. Styles from this library will appear on the **Styles** tab.
3. Tab **Styles**: enable the option near the required style name.

To finish the setup, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



You can use several report style libraries at a time and connect styles from them as required.

Connecting styles from libraries is also possible in the process of report creation. For details about creation of a report, see Section 5.2.3.3 on p. 1418.

5.2.2.3. Setup

5.2.2.3.1. Setting sheet format and fields

You can specify sheet format and its margins that will be used when printing the current report from the Data Proceeding Window.

In order to set up the report's page format, call the **Service — Parameters... — System — Reports — Table positioning — Page format command**. Select document sheet format from the standard format set or specify arbitrary sheet size in the displayed dialog.

In order to set up report's page margins, call the **Service — Parameters... — System — Reports — Table positioning — Page margins**. Enter field values or use counter to select them in the displayed dialog.



You can view the report with the current settings of sheet format and fields in the Data Processing Window in the page layout mode. Use the **Page Layout** button in the **View** bar of the Data Processing Window to enable this mode.

The control elements of this dialog are described in Section 9.1.12.3 on p. 1743.

5.2.2.3.2. Text Parameter Settings

You can set parameters of the headings' text and the report's table cells:

- ▼ for the text documents (see Section 9.2.4.7 on p. 1754)— setup is used when placing a report in the text document.
- ▼ for graphic documents (see Section 9.2.6.23 on p. 1835)— setup is used:
 - ▼ when placing a report in the graphic document;
 - ▼ when displaying a graphic document report in the Data Processing Window;
- ▼ for model documents (see Section 9.2.7.8 on p. 1865)— setup is used when displaying model report in the Data Processing Window.

The control elements of this dialog are described in Section 9.2.4.3 on p. 1752.



The **Alignment** parameter is an exception as it is specified in style settings for each column separately (for more info on style settings, see Section 5.2.2.2.3 on p. 1408).

In the setup dialogs described in Section 9.2.4.3 on p. 1752, you can specify only **style** alignment method parameters. To do this, select an alignment method (**right**, **left** or **center**) with non-zero indent.

The report can be created for the objects of one document and placed in another document. If text settings are different in these settings, report view in the Data Processing Window may differ from the view of this report inserted into a document.

By default, for text, graphic documents and model documents, similar report text parameters are set.

Table text name parameters are set up:

- ▼ for non-associative reports — in the table text parameters setup dialog (see Section 9.2.6.16.3 on p. 1818),

- ▼ for associative reports — in the report text parameters setup dialog (see Section 9.2.6.23.3 on p. 1835).

5.2.3. Creation of reports and working with them

5.2.3.1. Objects inserted in a report

Objects that can be selected for insertion in a report:

- ▼ in model
 - ▼ bodies;
 - ▼ components:
 - ▼ subassemblies;
 - ▼ parts and local parts;
 - ▼ standard parts;
 - ▼ model library components;
- ▼ in drawing
 - ▼ image inserts from views of other drawings;
 - ▼ insertions of fragments;
 - ▼ graphic macro elements — user-defined (created by the **Create Macro Element** command) and library (for example, inserted from the Standard Items Library);
 - ▼ models of associative view; they are placed in the report if model's view or one of the model's objects is specified in the Drawing Tree;
 - ▼ views; all objects contained in it are placed in the report (see above);
- ▼ in fragment
 - ▼ insertions of fragments;
 - ▼ macroelements.

5.2.3.2. Report creation and insertion commands

Report is created using the **Create Report** command. The command is available in graphics documents:



- ▼ model (**.a3d*)— report is created using the current model and placed in the selected in the command graphic or text document (new or existing);
- ▼ drawing (**.cdw*), fragment (**.frw*) — a report is created using current document objects and placed in the same document.

See Section 5.2.3.3 for more information about the command.



The **Insert Report** command is used to insert a report into a document.

The command is available in the graphic (**.cdw*, **.frw*) and text (**.kdw*) documents.

Once the command has been called, select the model document or a graphical document in the file opening dialog whose objects will be included in the report. Once the selected document is

opened, the report creation process is started as after calling the **Create Report** command (see Section 5.2.3.3). After report placement in the document, the command is automatically terminated.

5.2.3.3. Report creation procedure



In order to create a report on objects' properties in a graphical document or model, call the **Create report** command.

Perform the following actions after the command has been called.

1. Using the switches of **Method** group specify objects the data about which must be transferred to a report.

By default, the switch is active



- ▼ in the model document — **By all objects up to specified level**; if this option is used, the report will contain all components of the first level;



- ▼ in a drawing — **By current view**, if this option is enabled, the report will include all objects of the current view.

Object selection for a report is detailed in Section 5.2.3.3.1 on p. 1419.

To automatically add objects with definite properties to a report, specify filtering conditions (see Section 5.2.3.3.4 on p. 1421).

2. Select a report style from the **Style** drop-down list that contains all connected styles.

If there is no the style you need in the list, click the **Other...** row. Another dialog will appear on the screen where you can choose a style from the library of styles and connect it. Working with the dialog is described in Section 5.2.2.8 on p. 1415. The style selected in the dialog will automatically become selected.

Press the **Customize...** button to view or edit the selected style. Style setup is detailed in Section 5.2.2.3 on p. 1408.

3. You can enable automatic assignment of numbers to report tables. By selecting the name of the numbering group in the **Numbering** list (see Section 3.9.1 on p. 1277).
4. Select the continuation of command operation.



The **Data Proceeding Window** switch is active by default. It means that after pressing the **Create Report** button you will switch to the Data Proceeding Window. This window allows to view and edit a report, make advanced appearance settings and save a report to an independent file (**.cdw*, **.frw*, **.kdw*, **.tbl*, **.txt*, **.ods*, **.xls*) or print it. From the Data Processing Window you can go to table placement in a document.

If the **Data Processing Window** switch is not active, after pressing the **Create Report** button you will switch to the table placement in the document.

If the **Associative report** option is enabled, the report's table will be created which is associatively linked to the document based on whose documents it has been obtained. If the option is disabled, the report is created without the associative link. For details, see Section 5.2.3.8 on p. 1433.

If the **Data in the form of links** option is enabled, links to values of objects' properties will be automatically formed in the report. If the option is disabled, the report consists of simple text. For details, see Section 5.2.3.7 on p. 1432.



Associativeness and data display as links settings are only applicable when placing report n a graphical document.



If the report is created for a model, the options of **Associative table** and **Data in the form of links** are available on the Properties bar only if the model is saved to file on disk.

In a model document the report is placed in the new document when the option **To New Document** is enabled. New document type is selected from the **New Document** list of the Properties bar. If the option is disabled, the report will be inserted in the earlier created document. Report table location parameters are specified on the **Assembly** tab (see Section 5.2.3.3.3 on p. 1421).



5. Click the **Create Report** button on the Special Control Panel.

When the **Data Processing Window** switch is active, the Data Processing Window containing the report will be displayed. Customize the report. Working in the Data processing window is detailed in Section 5.2.3.4 on p. 1423.

If the switch is inactive, the screen displays the document selection dialog or the standard file open dialog. Select a document. Specify report insertion place in the opened document. A name can be entered for tables located in graphic files — see Section 4.2.2.1.3 on p. 1364.

After the report is inserted using the **Create Report** command or the Data Processing Window is closed, you will automatically return to the report creation process. After the report is inserted by the **Insert Report** command, the process finishes. For more information on the report creation and insertion commands, see Section 5.2.3.2 on p. 1417.

You can create several reports per one command call and place each of them in several documents or print them.



To change all objects press **Select Other** button on the Special Control Panel and re-select objects.



After finishing work with reports, press **Interrupt Command** button in the Special Control Panel.

5.2.3.3.1. Object selection methods

To select the right object selection method, activate the switch in the **Method** group on the Properties bar. Then, depending on the method, perform required actions.

In a model document



If **At All Objects to Specified Level** switch is active, components and bodies up to the specified level, obtained by operations in the assembly, are automatically included into an assembly report.

By default, first-level objects are selected.

In the **Level** field set the level number being the inclusive limit, to which auto selection of objects is made.



If **At All Levels** switch is active, all-level components and bodies obtained by operations in the assembly are automatically included into an assembly report.



If the **By specified objects** switch is active, objects must be selected manually by clicking them with mouse. Specify the objects. If necessary, you can specify the model itself. The selected objects are highlighted. To cancel the item selection, click it again.



To select bodies constructed by operations in a part or components included into a part, it is necessary to use the **By specified objects** method.

In drawing



If the **By Current View** switch is active, all current view objects are included into a report.



If the **By All Views** switch is active, all drawing view objects are included into a report automatically.

If there are several views from one model in a document, the information is not duplicated—only main view information is displayed in the report. The main view is the view which was used to obtain all other views or the view which was created as the first one.



If the **By specified objects** switch is active, objects must be selected manually by clicking them with mouse. Specify the objects.

The **Model Reading Level** option is used to enter and select a level from the list to which (inclusive) objects are selected from assemblies in the drawing's associative views. When the option is enabled, the level selection list is available.



In the associative drawings, if **By current view** and **By all views** methods are used, the report will contain the bodies obtained using operations in the assembly and assembly components. To select bodies constructed by operations in a part or components included into a part, it is necessary to use the **By specified objects** method.



Objects can be selected before calling the command. In this case, the **By specified objects** switch is active, and the objects are considered as manually selected. The view in the associative drawing is selected if it had been highlighted as a whole. For this purpose, for example, you can click its dimensional frame.

The selected objects are highlighted. To cancel the item selection, click it again.



When customizing **fragment** report, the **Method** group is not displayed in the Properties bar. All objects are automatically included to a report.

Object type is customized on the **Objects** tab of the Property bar (see Section 5.2.3.3.2 on p. 1420). If the type is disabled, the objects of this type will be come unavailable for selection by **By specified objects**, and previously selected objects of this type are deselected.

5.2.3.3.2. Selection of object types

The setting allowing to specify type of objects to place in the report is performed on the **Objects** tab of the Properties bar (Fig. 5.2.5).

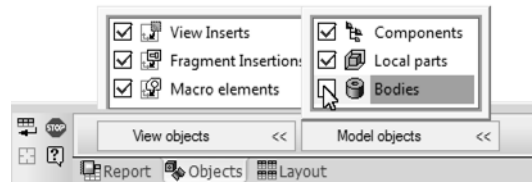


Fig. 5.2.5. **Objects** tab of the Properties bar in a drawing

The **View Objects** and/or **Model Objects** panels that contain the list of the object types are located on the tab.

Enable the option located to the left of the report's name for this type of objects to participate in the report creation.

If a certain type's option is disabled, these objects will be excluded from those selected irrespective of the objects' selection result using the **Method** group (see Section 5.2.3.3.1 on p. 1419).

5.2.3.3.3. Report Assembly

The setting that serves to select the method of positioning tables on a sheet is made on the **Assembly** tab of the Properties bar (Fig. 5.2.6).

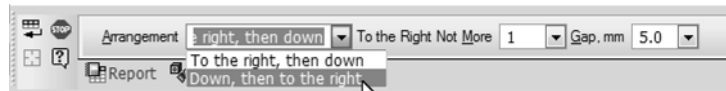


Fig. 5.2.6. Layout **tab of the** Properties bar

Choose the method in the **Assembly** list.

- ▼ When you select the **To the Right then Down method**, tables are placed in horizontal rows. In the **To the Right not More** field set the maximum number of tables in a horizontal row.
- ▼ When you select the **Down then to the Right method**, the tables are placed in vertical rows. In the **Down not More** field set the maximum number of tables in a vertical row.

Element **Gap, mm** makes it possible to enter or select from the expandable list the distance between borders of tables in rows.

Set the distance between table borders in rows in the **Gap, mm** field or select from the drop-down list.

Report placement parameter settings remain effective till the end of the document working session.

5.2.3.3.4. Customizing object filtering conditions by properties

Filtering by object properties allows to automatically select and insert in the report the objects with the properties meeting the specified conditions. Filtration is applied to objects selected with the help of **Method** group or **Type** tab of the Properties bar (see Sections 5.2.3.3.1–5.2.3.3.2 on p. 1420).

Filtering by object properties is available in a model document. You can perform row filtering for graphic object and model reports in the Data Processing Window (see Section 5.2.3.5.2 on p. 1426).



Filtering mode control is performed using the **Filter Objects by Properties** button on the Special Control Panel. The pushed button means the filtering mode is enabled, the released — that it is disabled. The button is available if filtering conditions are specified.



To specify and view filtering criteria, press the **Customize Object Filter by Properties** button. After pressing the button, the **Filtering Criteria** tab appears on the Properties bar.

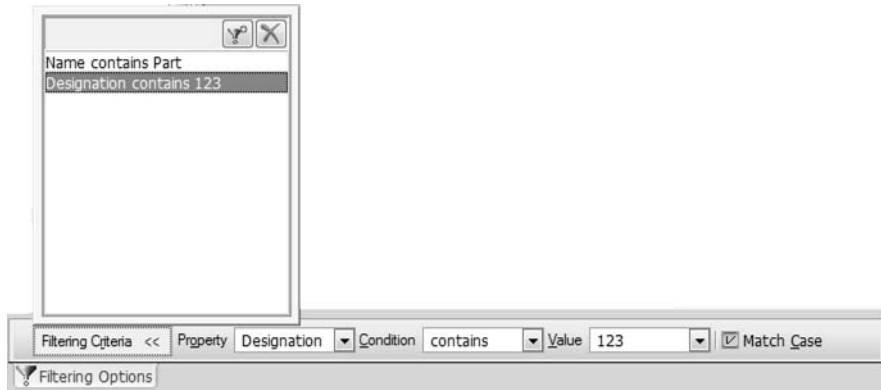




Fig. 5.2.7. **Filtering Criteria** tab when selecting objects in a model document

Filtering conditions are displayed in the window of the **Filtering Criteria**. Every condition consists of three parts: *<Property>* *<Condition>* *<Value>*. If the window is empty, the conditions are not specified.

The tab controls are described in Table 5.2.5.

Tab. 5.2.5. Parameter Setting Window Controls

Element	Description
Filtering Criteria	Panel that allow to form filtration conditions. The window serves to display the filtering condition list.
 Add Condition	The button allowing to add a condition to the list. After pressing the button, the first part of the filtering condition appears in the window—default property <i>Designation</i> . This property condition and value are not specified.
 Delete Condition	The button allowing to delete a selected condition or property.
Property*	Drop-down list from which you can select filtration property. The properties connected to a document are available in the list.

Tab. 5.2.5. Parameter Setting Window Controls

Element	Description
Condition*	The drop-down list where you can select the condition for the current filtration property. Selected condition is displayed on the Filtering Criteria panel.
Value*	This field is used to meter a property value. The value can be also selected from the drop-down list that contains previously used values.
Match Case	The option allowing to match case — upper-case or lower-case letters in the value text entered in the Value field. If the option is enabled, the case will be matched. If the option is disabled, the case will not be matched.

* This control element is available if field the filtering condition window contains at least one property.

To specify filtering conditions, do the following.



1. Press the **Add Condition** button located on the **Filtering Criteria** panel. The field will display the default filtering property option.
2. Expand the **Property** list and select filtering property. Set filtering parameters using the **Condition** and **Value** controls.
The full condition appears on the **Filtering Criteria** panel. For instance, the *Name Contains Flange* condition means that the report will include the rows with word 'flange' in their names.
3. To specify the next condition, repeat the actions described in items 1 and 2.



If the **Filtering Criteria** panel contains several conditions, the report will include the objects meeting at least one of these conditions.

To edit a filtering condition, select it in the **Filtering Criteria** panel and modify the desired parameter.



Confirm filter creation by pressing the **Create Filter** button in the Special Control Panel of the tab.

Filtering criteria will be effective till the end of the KOMPAS-3D working session.

5.2.3.4. Data Processing Window

5.2.3.4.1. General Information

The Data Processing Window serves to view a report, to edit and to additionally customize report appearance. The report customized in the Window can be printed, inserted in a document or saved to a separate file.

The data processing window appears on the screen in the following cases:

- ▼ when creating a report, it appears after you click the **Create Report** button on the Special Control Panel of the **Create Report** command (with the **Data Processing Window** switch pressed on the Properties bar).
- ▼ when editing the associative report, it appears after calling the **Edit in the data processing window** command from the context menu of the report selected in the report (or after double-clicking its dimensional frame).

The Window contains a completed report table in the form specified in the Property Bar of the **Create Report** command and in the **Objects** tab with either enabled or disabled object filtering mode (for models) specified on the Special Control Panel. When editing an associative report, changes made in the table in the document's field are also displayed in the Data processing window.

Report table contains selected objects in the rows and the set of these objects properties in the columns. Report appearance corresponds to the report style.

By default, data is displayed in the report's lines as links to objects' properties (see Section 5.2.3.7 on p. 1432).

The report row can consist of several table rows. This is due to the fact that cell text length can exceed the column width specified in the report style.

By default, the report displays in grouped form (see Section 5.2.2.2.5 on p. 1412).



The header text and table cell parameters of the Window conform to the parameters set for the current document by default.

Text parameter setup is detailed in Section 5.2.2.3.2 on p. 1416.

Data proceeding window contains the following panels.

- ▼ Properties bar (Fig. 5.2.8).

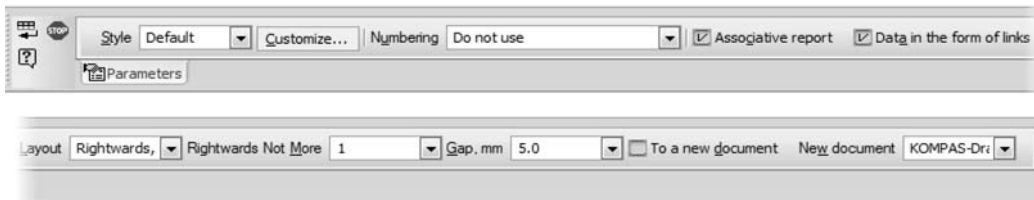


Fig. 5.2.8. Properties bar of the Data proceeding window during creation of a document.

Panel control elements serve to customize the current report parameters.

- ▼ Compact bar that contains several toolbars.



Fig. 5.2.9. The Compact Bar in the Data Processing Window. **Report Table** toolbar is active

- ▼ The **Report Table** bar includes the command buttons to finalize the report— to group, filter, select rows to place them in the report, etc., as well as commands to insert the report in a document or save it in a separate file (Fig. 5.2.9).

- ▼ The **Formatting** toolbar serves to call the commands for text formatting in the report table cells (see Section 4.1.2.5 on p. 1297).
- ▼ The **Insert into Text** bar serves to call the commands for inserting various objects into the report text (see Section 4.1.2.11 on p. 1308).

From the Data Processing Window you can

- ▼ save report to a separate files (see Section 5.2.3.5.13 on p. 1431);
- ▼ print report (see Section 5.2.3.5.14 on p. 1431);
- ▼ place report in the document (see Section 5.2.3.5.15 on p. 1431).
- ▼ when editing an associative report, you can return to the document in which it is placed.



To finish work in the Data Processing Window, click the **Interrupt Command** button on the Special Control Panel. You can also close the Window with the **Close** button in the upper right corner.

5.2.3.4.2. Modes of work in the Data Processing Window.

There are three display modes— normal, page layout mode and report display mode without document border and empty rows.

Z



Fig. 5.2.10. The **Modes** bar in the Data Processing Window

You can easily switch between the modes by using buttons from the **View** panel or commands from the **Modes** menu. If the button is enabled, it indicates the current mode.



After the Data Processing Window is opened, the report table will be displayed in the normal mode. In this mode, the table lists all objects together without pagination, and the document border is not displayed. In the normal mode all report settings are available.



To display the report with pagination and margins, switch to page layout mode. To switch to this mode, press the **Page Layout** button on the **Modes** panel.

In this mode you can modify the report style and placement parameters located in the **Parameters** tab of the Window Property Bar. In this mode you can also cancel row filtering and you can control grouping. You cannot use other table editing methods.



In the **Page Layout** mode the report display without document border and empty rows is available. To do this, press the **Without Border and Empty Rows** button.

The **Current Status** toolbar serves to navigate through the report (Fig. 5.2.11). Its fields display help information on numbers of the current page, row and quantity of report rows.

To move to another page or row, enter its number in the **Current Page** or **Current Row** field and press the **<Enter>** key.



Fig. 5.2.11. The **Current Status** bar in the Data Processing Window

If the **Current Status** bar is located vertically, press the respective button to access the field.

5.2.3.5. Methods to working in the Data Processing Window

5.2.3.5.1. Setting the current parameters in the Data Processing Window

You can create a report either in grouped or ungrouped form.

By default, the report is grouped. This means that objects with similar properties in a report appear in one resulting row.



An indicator of a grouped report is the pressed **Group** button in the **Report Table** toolbar or in the **Table** menu.

Customization of grouping parameters is in the report style.

To view or modify grouping parameters, press the **Customize...** button in the Property Bar of the Data Processing Window. To customize grouping, use the **Grouping** row cells. For details see Section 5.2.2.2.3–5.2.2.2.5 on p. 1412.



Having modified the style, call the **Rearrange Table** command.

The report with modified grouping parameters will be displayed in the Data Processing Window.

Depress the **Group** button to get a full list of objects in the report.



When you cancel grouping, row filtering is canceled, too. For information about row filtering, see Section 5.2.3.5.2.

5.2.3.5.2. Filtering Rows

You can apply filtering to the table displayed in the Window— leave in a report only rows with the column values meeting the specified conditions or containing a definite set of characters.



The **Apply Filter** command serves for filtering report rows. To call the command or view filtering conditions, press the button in the **Report Table** bar or select the name from the **Table** menu. The **Filtering Criteria** tab appears on the Properties bar (Fig. 5.2.12).

Filtering conditions are displayed in the window of the **Filtering Criteria** panel. Every condition consists of three parts: *<Column name>* *<Condition>* *<Value>*. If the window is empty, the conditions are not specified.

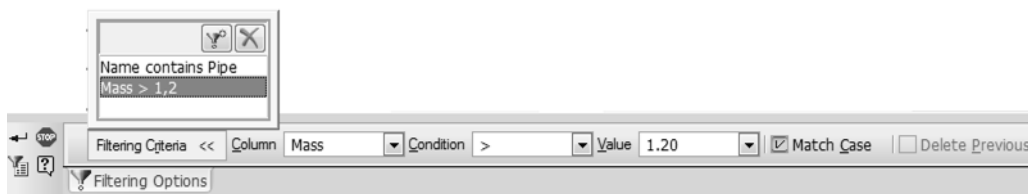


Fig. 5.2.12. The **Filtering Criteria** tab during row filtering

Specify filtering conditions using tab control elements. For this, do the following.



1. Press the **Add Condition** button located on the **Filtering Criteria** panel. After pressing the button, the first part of the filtering condition appears in the window— default column *Designation*. Condition and value of the property displayed in this column are not specified.
2. Expand the **Column** list and select filtering property. The **Column** list contains column names of the current style.
Set filtering parameters using the **Condition** and **Value** controls.
The full condition appears on the **Filtering Criteria** panel. For example, condition *Length, m < 1.2* means that the report will include the rows with values less than 1.2 in the *Length, m* column.
3. Enable the **Match Case** option to match case— upper-case or lower-case letters in the value text entered in the **Value** field.
4. To specify the next condition, repeat the actions described in items 1–3.



If the **Filtering Criteria** panel contains several conditions, the report will include the objects meeting at least one of these conditions.



If you want to save the customized filtering parameters till the end of the document working session, call the **Remember Filtering Conditions** command on the Special Control Panel.



To perform filtering, press the **Create Object** button on the Special Control Panel.

A filtered table will be displayed.

To edit a filtering condition, select it in the **Filtering Criteria** panel and modify the desired parameter.

You can apply new filters to the already filtered table. To do this, recall the **Apply Filter** command. In this case, the **Delete Previous** option is available in the Property Bar. If the option is enabled, previous conditions are ignored — filtering will be performed with new conditions solely. If the option is disabled, new conditions will be applied in addition to the previous ones. To apply filtering, press the **Create Object** button again.



To cancel row filtering made in the Data Processing Window, call the **Delete Filters** command. For this, press the button in the **Report Table** bar or select the name from the **Table** menu. The command is available if the **Apply Filter** command have been executed.

5.2.3.5.3. Row Selection

Report row selection can be required both when editing (deleting, copying, inserting, moving rows within a table) and when creating a report from separate rows.

You can highlight rows by the following methods.

- ▼ To select a row, click it with the mouse.
- ▼ To select several rows, click the mouse on them holding the *<Ctrl>* key.
The repeated mouse button pressing on the selected row cancels its highlighting.
- ▼ To highlight range of rows, press and hold the *<Shift>* key and left-click on the start and end row of the range. If the range is started from the first row, with the pressed *<Shift>* key, left-click only on the end row.

Also for selection in the required direction you can use *<Shift> + <↖>/<↗>* key combination.

To shift the selected limits of the row range, click on the new end row with the *<Shift>* key pressed.

To deselect all rows, click the mouse on an empty row.



Number of selected report rows is displayed in the **Current Status** bar in the **Selected Rows** field.

Rows can be selected in the normal report display mode (see Section 5.2.3.4.2 on p. 1425).

5.2.3.5.4. Hiding rows

You can hide some of the rows so that they do not appear in the report. To do this, select rows which must be displayed in the report.



Call the **Hide All Rows Except For Selected** command to view the report compiled from selected rows. To call the command, press the button in the **Report Table** bar or select the name from the **Table** menu.

After the button is pressed, all rows except for the selected ones are not displayed.



Report editing is impossible in the row hiding mode.

To restore the report state with all rows displayed, release the **Hide All Rows Except For Selected** button in the **Report Table** bar or recall the command from the **Table** menu.

5.2.3.5.5. Inserting rows

You can insert row anywhere in a table.

To perform the operation, select a row below which row must be inserted.



Call the **Insert a Row Below** command To do this, press the button in the **Report Table** bar or select the name from the context menu or from the **Table** menu. You can also use *<Insert>* key.

After command calling, the row is inserted. Text editing mode in which you can enter the text is launched simultaneously. For more on the report text editing mode see Section 5.2.3.5.10 on p. 1429.

To leave the line blank, click the mouse on any line after the insertion and press *<Ctrl> + <Enter>* or *<Esc>*.

5.2.3.5.6. Copying rows

You can insert a copy of the selected row in a report.

To perform the operation, select the row to be copied.

Call the **Copy Row** command from the context menu or from the **Table** menu.

After calling the command, the copy is added below the copied row.

5.2.3.5.7. Deleting rows

You can delete selected rows.



To do this, after selecting rows call the **Delete Row** command by pressing the button in the **Report Table** bar, or by selecting the name from the context menu or from the **Table** menu. You can also use *<Delete>* key.

5.2.3.5.8. Moving rows up and down



You can move a selected row up or down.



To move a row, call the **Move Row Up/Down** command,

You can call the commands by pressing the button in the **Report Table** bar, by selecting the name from the context menu or from the **Table** menu. You can also use the key combination *Ctrl* + *<^>*/*<↓>*.

You can move range of selected rows in the same way.

5.2.3.5.9. Moving rows to the beginning of the page



You can move rows, starting with a selected one, to the beginning of the next page.

To do this, select a row from which new page must be started and call the **From New Page** command by pressing the button in the **Report Table** bar, by selecting the name from the context menu or from the **Table** menu.

To return moved rows to the source page, select the row to which the **From New Page** command was applied. In this case, the button to call this command will be pressed. Then release the **From New Page** button.

5.2.3.5.10. Editing text in a report

You can modify a report row text or merge row cells.

In order to enter the text edit mode, perform one of the following:

- ▼ double click in a row;
- ▼ select the line and call command **Edit Line** from the context menu or from the **Table** menu;
- ▼ press *<Enter>*.

This command will display Edit Line Property bar in text mode.

In the text edit mode, the controls of the **Formatting** and **Insert into Text** bars are available.

Perform the text-editing actions.

In order to leave the edit mode and confirm changes, do one of the following:



- ▼ click the **Create Object** button on the Advanced Control Panel.
- ▼ click any place of the table outside the edited line;
- ▼ press *<Ctrl>*+*<Enter>*.

To exit editing mode without confirming the changes, press the *<Esc>* key.



To cancel text editing, select the edited row and call the **Restore Text Part** command from the context menu.

If the **Data in the form of links** option is enabled in the Property bar, links to objects' properties are displayed in lines. They are non-editable. You can enter text prior to or after the link in the cell or delete the whole link, if necessary.



In order to convert data in the form of links to ordinary text, disable the **Data in the form of links** option and call the **Rebuild table** command. The option is available when creating a report only.

For more information about working with data in the form of links, see Section 5.2.3.7 on p. 1432.

5.2.3.5.11. Merging cells in a row

You can merge adjacent cells in a row.

For this purpose, enter the text edit mode.



Select the cells to be merged and click the **Merge cells** button in the Property Bar or call the respective command from the **Table** menu.

Confirm your editing.

Entering the text edit mode and exit from it are described in Section 5.2.3.5.10.



To cancel cell merging, select the edited row and call the **Restore Text Part** command from the context menu.

5.2.3.5.12. Searching text in a report

You can find the row containing the specified text.



To perform search, call the **Find...** command from the **Editor** menu.

After calling the command, the **Find** tab appears on the Properties Toolbar. Customize search parameters using **Find** tab control elements

To do this, enter the desired text in the **Text** field. It can also be selected from the list that contains previously used text fragments.

If you need to find the text contained in the values of some property, enable the **In Column** option and select this property from the **Column** list.

If you want to find cells that only contain the specified text, enable the **Whole Cell** option.

If you want to match character case, enable the **Match Case** option. In this case, the search will distinguish lower-case and upper-case characters. Thus, for instance, 'GOST' and 'Gost' will be perceived as different words.



Select the search direction using the **Direction** group switches.

- ▼ if the **End** switch is active, the search is performed from the beginning to the end of the document;
- ▼ if the **Home** switch is active, the search is performed in backward direction;
- ▼ if the **Find All** switch is active, all rows with the desired text will be found in a report.



To start search, click the **Execute** button on the Special Control Panel.



When the **End** or **Home** switch is active, the row with the desired text will be selected. Repeat search by clicking the **Apply Search** button. Each time the button is clicked, there is selected one row found in the specified direction.

When the **Find All** switch is active, all found rows are selected.



If the desired text is not found, the respective message will be displayed on the screen.



You can also activate the **Find...** command in the text edit mode from the **Editor** menu or by pressing **<Ctrl> + <F>** key combination. In this case text is searched in the table within the current cell.

Customization of cell search parameters is detailed in Section 4.1.2.9 on p. 1305.

5.2.3.5.13. Saving to a separate file



Report data can be saved to separate files.

To save the report, call the **File — Save As...** command or press the **Save Current Report As...** button in the **Report Table** bar.

Upon pressing the button, a standard file save dialog opens.

The report can be saved as a document of one of the following types.

- ▼ KOMPAS-Drawings (the **.cdw* file),
- ▼ KOMPAS-Fragments (the **.frw* file),
- ▼ KOMPAS-Text Documents (the **.kdw* file),
- ▼ KOMPAS-Tables (the **.tbl* file),
- ▼ Text Files (the **.txt* file),
- ▼ ODF Spreadsheets (the **.ods* file),
- ▼ Microsoft Excel Spreadsheets (the **.xls* file).

5.2.3.5.14. Print preview and printing

Report in the current view can be printed.

Customization of sheet format and fields is described in Section 5.2.2.3.1 on p. 1416.



You can view report placement on the pages in the Data Processing window in the **Page Layout** mode (see Section 5.2.3.4.2 on p. 1425).



You can also use data processing window before KOMPAS-document printing. Press the **Preview** button on the **Standard** panel. The window in which you can customize different print parameters will appear on the screen (see Section 8.2.1.2 on p. 1639).



To print a report, press the **Print** button in the **Standard** bar or call the **File — Print command**. You can also use the key combination **<Ctrl> + <P>**.

The dialog similar to the text document print setup dialog will be displayed on the screen. Set print parameters.

Document printing is detailed in Section 8.1 on p. 1631.

5.2.3.5.15. Placing a report in a document

You can place all or only selected report rows in a document.



To transfer to a document all rows displaying in the Window, call the **File— Transfer to Document— Report** command or press the **Transfer Report to Document** button on the **Re-**

port Table bar. You can also call the **Create Report** command on the Special Control Panel of the Data Processing Window.



To transfer selected rows to a document, call the **File — Transfer to Document — Selections** command or press the **Transfer Selections to Document** button on the **Report Table** bar. The command is available if the report has selected rows.

If the **Hide All Rows Except Highlighted** button is pressed, then a report is in the selected row preview mode. Viewing of the report compiled of selected rows is detailed in Section 5.2.3.5.4 on p. 1428.

This command will display the dialog to select a document or standard dialog to open files.

Once the selected document is opened, the system will automatically proceed to placing tables in it. In a model document the report is placed in the new document when the option **To New Document** on the Properties bar is enabled.

Specify table anchor point position with the cursor or by entering coordinates in the Property Bar.

5.2.3.6. Names and numbering of report tables

For report tables put in a graphic documents, names can be entered (see Section 4.2.2.1.3 on p. 1364) and auto numbering can be enabled.

Auto numbering means the table is assigned a number link to the number in the numbering group. To do this, add the table to the group using one of the methods:

- ▼ when creating or editing a report — by selecting the group from the **Numbering** list located in the Properties bar;
- ▼ after the report is put in the document — by adding to the **Numbering** window called from **Service — Numbering**.

Techniques for automatic numbering of objects are described in details in Section 3.9.1 on p. 1277.

5.2.3.7. Data in form of links

In reports that are placed in graphical documents, you can display the contents of report's lines in form of links to properties of values in the source documents. Such links are created automatically if the **Data in the form of links** option is enabled in the Property bar of the **Create report** command (see Section 5.2.3.3 on p. 1418). They are displayed in the report in the color that is prescribed for text links (see Section 9.1.3.5 on p. 1681).

You can enter ordinary text in a cell in front of or after such link: additional text, designation, special symbol, and so on.

It is possible in the following modes:

- ▼ in a document, in the table editing mode (see Section 4.2.1.1 on p. 1349);
- ▼ in the Data processing window, in the text editing mode (see Section 5.2.3.5.10 on p. 1429);

<i>Accessoires</i>	regular text
<i>Latch Dy 50 in set</i>	
<i>Latch Dy 150 in set</i>	
<i>Flange Dy 150</i>	

data as a reference text regular text

Fig. 5.2.13. Displaying data in the report as links



It is impossible to manually edit or explode data in the form of links. Once the properties' values are modified in the source document, you can update the data in the report using the **Rebuild** command; in the associative report, you can also use the **Update table** command (see Section 5.2.3.8.2 on p. 1435).

Data in the form of links in the report can change color in the Data processing window or in the document. It means that links should be updated, or links have become invalid, i.e. their link to the source document or object is lost.

In order to update links, rebuild the report. In order to restore the relation, remove the reason of its break.

5.2.3.8. Associative reports

You can create reports in which the associative link is maintained with the document on the basis of whose objects it is created.

The associative link will be formed, if the **Associative report** option was enabled during creation of the report in the Property bar.



The associative report should be placed in a graphical document. If placed in a text document, the associative link between the report and the source document will not be created.

The associative report has the following differences from a simple report (created with the **Associative report** option disabled).

- ▼ In the associative report, permanent link to the source document is maintained. The source document is the document whose objects were used for creation of the report. All changes taking place with respect to objects in the source document and related to the report are transferred to the report. For example, it will be automatically updated, if objects are added to the source document taking account of the method of specifying objects in use.

The report will be up-to-date due to regular updates (see Section 5.2.3.8.2 on p. 1435). If necessary, the user can explode the associativity at a certain stage of working with the report (see Section 5.2.3.8.3 on p. 1436).

- ▼ The associative report is a single object that may include several tables. It is displayed in the document in an additional external dashed frame (Fig. 5.2.14). A ready report can be edited both in the Data processing window or in the document in which it is placed (see Section 5.2.3.8.1 on p. 1434).

<i>Cold water route</i>			
<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	<i>Fastener Dy 50</i>	<i>11</i>	<i>70,9003</i>
	<i>Flange Dy 50</i>	<i>4</i>	<i>5,0697</i>
	<i>Gasket Dy 50</i>	<i>4</i>	<i>0,0726</i>
	<i>Pipe Dy 50</i>	<i>34</i>	<i>169,5397</i>
	<i>Pipe Dy 150</i>	<i>22</i>	<i>957,4884</i>
	<i>Pump</i>	<i>2</i>	<i>8598,9696</i>
	<i>T-branch Dy 150</i>	<i>4</i>	<i>21,0098</i>

Fig. 5.2.14. Associative report

Context menu of the associative report contains both typical commands for working with an object, for example, cut, copy, create hyperlinks and so on, and report editing commands in the Data processing window, source document opening, names entry, update (Fig. 5.2.15).

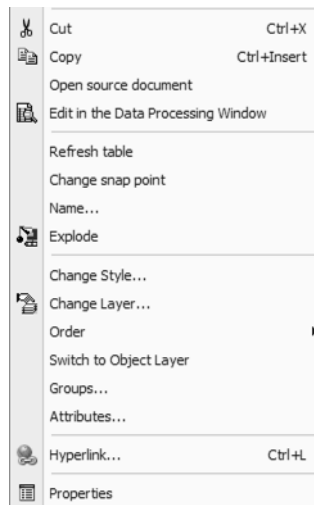


Fig. 5.2.15. Context menu of the associative report



Peculiarities of copying should be noted with respect to an associative report created by objects of the current graphical document. To keep the associativity of a report's copy, we recommend to use copying with the mouse method (see Section 3.4.1.1.2 on p. 1114).

5.2.3.8.1. Editing

The following methods of the associative report editing are supported.

- ▼ Editing objects in the source document.

Call the **Open source document** command from the context menu of the report selected in the document. Once the command is called, the document, for which the report is obtained, will appear on the screen. Modify objects in the source document, if necessary. The changes will be reflected in all associative reports that had been created for the document.

Update these reports in documents (see Section 5.2.3.8.2 on p. 1435).

- ▼ Report editing in the Data Processing Window.



Call the **Edit in the Data processing window** command from the context menu of the report selected in the document. The command can also be called by double-clicking the external frame of the report.

Once the command is called, the report is displayed in the Data processing window mode. Controls for changing the report's style, assembly, cells' contents and so on will be available in it.

Perform the same steps that are performed when creating a report (see Section 5.2.3.5 on p. 1426).



In order to confirm changes, click the **Create Report** button on the Special Control Panel.



When editing a report, it is impossible to control the method of displaying the **data in the form of links**, cancel the report's associativity, change location parameters.

- ▼ Changing the report's snap point.

Call the **Change snap point** command from the context menu of the report selected in the document. Once the command is called, the process is launched during which you can choose another snap point.



In order to confirm changes, click the **Create Object** button on the Special Control Panel.

- ▼ Editing the report's table in the document in which it is placed.

Double-clicking the report (except for the dotted frame) makes it possible to enter the table editing mode. This mode supports all tools for handling tables (see Section 4.2.1.1 on p. 1349).



Report editing performed in the document's field will be canceled after the next rebuilding of the report in the Data processing window, for example, when changing style. In this case, the report will be updated and the changes made will not be applicable.

When editing the report, first of all, it is recommended to perform operations related to its rebuilding or update, and then edit the text in the Data processing window or document.

5.2.3.8.2. Updating tables

Updating the tables' contents is required for associative reports if data in the source document has been modified. If a table is crossed in a document, it means that it is not up-to-date (Fig. 5.2.16).

In order to update the report's data, call the **Update table** command from the context menu of the report which is selected in the document.

Once the command is called, data in the report will be replaced by data from the source document, and the table will no longer be crossed.

<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	Capacitor	.6	3901,2439
	Fastener Dy 50	11	70,9003
	Flange Dy 50	4	5,0697
	Pipe Dy 50	34	169,5397
	Pipe Dy 150	22	957,4884
	Pump	2	8598,9696
	T-branch Dy 150	4	21,0098

Fig. 5.2.16. Report with data modified in the source document



The data, which was manually edited when working the report, will not be updated.

When updating, the method of object selection specified in the report creation process will be applied.



Tables are also updated when a graphical document, in which the report is located, is being rebuild by calling the **Rebuild** command.



A report update cancels settings made in the Data processing window – line insertion, filtering, grouping, line moving, and so on.

5.2.3.8.3. Exploding the associativity of a report



The **Explode** command makes it possible to cancel the associativeness of a report, i.e. remove the link between it and the source document. You can call the command from the context menu of the report selected in the document or from the **Editor** menu.

Once the command is called, the associative report will be converted into a simple one, for which table editing methods are available (see Section 4.2.1.1 on p. 1349). The report is displayed without its frame and name if these were assigned.

If the report includes several tables, each table becomes an individual object.

Once the command is called, links to the values of properties will remain in the report.

5.2.3.9. Examples of report creation

5.2.3.9.1. Simple report

We recommend to do the following to create a product composition report.

1. Open the file — an assembly or drawing — on which you need to create a report. If this is a non-associative report, objects to be included in the report (nodes, parts, etc.) should be graphical macro elements and fragment insertions.



Call the **Create Report** command.

2. In the **Method** group, select:



- ▼ for an assembly — **By all levels** to get a report that lists all parts within assembly units.

- ▼ for a drawing — **By all views** to get a report that lists objects from all drawing views.



3. Press **Customize...** button.

Customize the report style in the opened dialog **Report Style Parameters**.

- 3.1. Add two columns (to the **By Default** style), select designations for them — *Position* and *Object Type*.
- 3.2. Customize sorting parameters by clicking on the **Sorting** field.

In the data sorting dialog, set the first level — **According to List** for the *Object Type* property, the second and the third levels — **A-Z** for the *Designation* and *Name* properties, respectively.

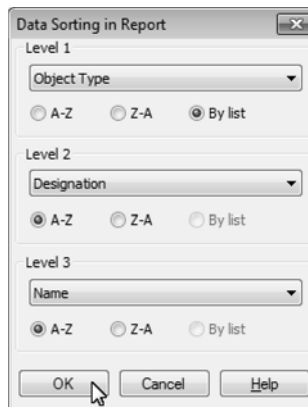


Fig. 5.2.17. Example of sorting setup

Note that the **Object Type** property is added to the report style for sorting setup solely. Hide it in the style using the **Show** option.

- 3.3. Set up the rest style parameters — column width, alignment and others mentioned in the dialog (Fig. 5.2.18).

Enter name in the **Style Name** field.

Save the created style to the style library by pressing the **Save to Library As...** button.

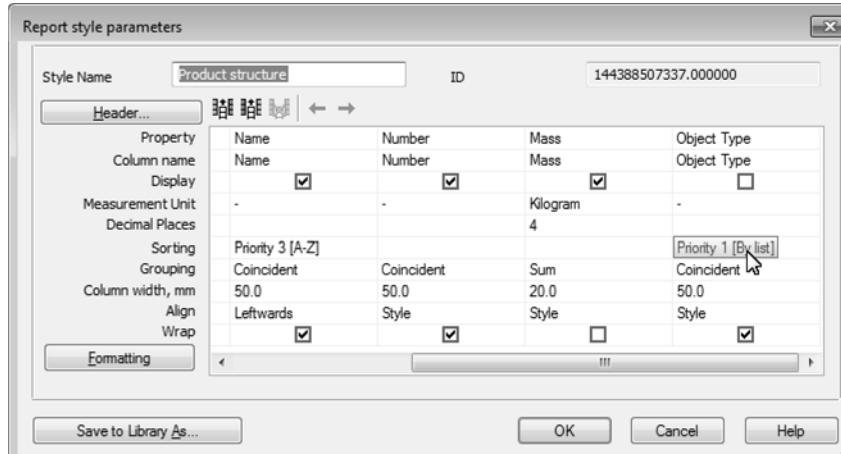


Fig. 5.2.18. Example of style setup



3.4. Go to the Data Processing Window. For this, press the **Create Report** button on the Special Control Panel while holding the **Data Processing Window** switch.



3.5. Create the report without the associative link with the document and without links. For this purpose, disable the **Associative report** and **Data in the form of links** options in the Property bar of the Data processing window.



3.6. Update the report. For this purpose, click the **Rebuild table** button on the **View** bar in the Data processing window.

RefNo	Designation	Name	Number	Mass	Object Type
078505.00101.00		Shaft 078505.00101.00	1	9.2123	Part
078505.00102.00		Gear	1	0.522	Part
078505.00103.00		Gear 078505.00103.00	1	6.4811	Part
078505.00104.00		Gear Housing	1	12.9038	Part
		078505.00104.00			
078505.00105.00		Lid 078505.00105.00	1	1.9113	Part
078505.00106.00		Lid 078505.00106.00	1	6.9648	Part
078505.00107.00		Pushing	1	0.152	Part
		078505.00107.00			
078505.00108.00		Pushing	1	0.0735	Part
		078505.00108.00			
078505.00108.00		Pushing	1	0.0735	Part
078505.00110.00		Gland 078505.00110.00	1	0.0238	Part
078505.00111.00		Washer	1	0.0371	Part
078505.00113.00		Washer 8 N	8	0.0077	Part
078505.00114.00		Bolt M6x25	8	0.1268	Part
078505.00115.00		Screw M6x25	1	0.0713	Part
078505.00116.00		Bolt M12x40	9	0.4673	Part
078505.00117.00		Washer 12 N	11	0.0354	Part
078505.00118.00		Bolt M12x60	2	0.1395	Part

Fig. 5.2.19. Example report in the Data Processing Window

4. Edit the report using the commands of the Data Processing Window — insert empty rows if required, enter text, move or delete rows, etc. (Fig.5.2.19).



5. Save the report as a separate document using the **Save current report as...** command or place the report in the already existing document (current drawing) using the **Create report** command (Fig. 5.2.20).



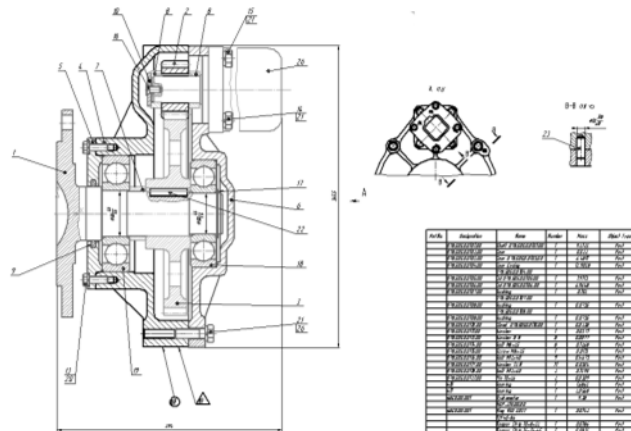


Fig. 5.2.20. Example of report placement in a drawing

5.2.3.9.2. Associative report

As an example, let us consider creation of an associative report, in which links are displayed in the form of properties, placing the report in a new fragment, validating links and associativeness of the report, as well as editing of the report, insertion of table names, exploding the associativeness.

The following procedure is recommended.

Creating a report



1. Open the file of the assembly to create a report on.

Call the **Create Report** command.



2. Choose the **By all levels** option in the **Method** group.

3. Enable the **Associative report** and **Data in the form of links** options in the Property bar, if these were disabled.

4. Enable the **To new document** option and choose the **KOMPAS-Fragment** variant in the **New document** list in the Property bar.

5. Customize style parameters as for a simple report (see Section 5.2.3.9.1 on p. 1436).

6. Begin placing the report in the document. For this, disable the **Data Processing Window** switch and click the **Create Report** button on the Special Control Panel.



7. Point the report's snap point in the fragment by the mouse.



8. The system will automatically return to the source document. In order to view the created report, go the fragment, in which it is placed.

<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	<i>Capacitor</i>	<i>6</i>	<i>3901,2439</i>
	<i>Fastener Dy 50</i>	<i>11</i>	<i>70,9003</i>
	<i>Flange Dy 50</i>	<i>4</i>	<i>5,0697</i>
	<i>Pipe Dy 50</i>	<i>34</i>	<i>169,5397</i>
	<i>Pipe Dy 150</i>	<i>22</i>	<i>957,4884</i>
	<i>T-branch Dy 150</i>	<i>4</i>	<i>21,0098</i>

Fig. 5.2.21. Associative report with data in the form of links

Associative modification of a report

9. In order to validate the report's associativity, go the source document using the **Open source document** command that should be called in the fragment from the report's context menu.
10. Add a new component to the assembly using any method and set its properties' values.
In the example, to *Pump* elements are added.

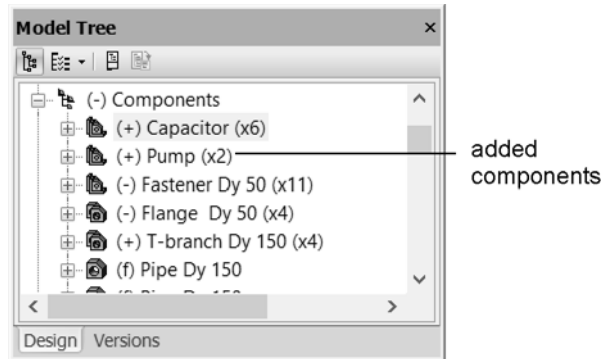


Fig. 5.2.22. Adding a component to the source document

11. Go to the document in which the report is placed. The report will be displayed as crossed in it, since the model's assembly in the source document has been changed.
12. Update the report. For this purpose, call the **Update table** command. After the update, the report will contain a new object — *Pump* components with preset properties.

<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	<i>Capacitor</i>	<i>6</i>	<i>3901,2439</i>
	<i>Fastener Dy 50</i>	<i>11</i>	<i>70,9003</i>
	<i>Flange Dy 50</i>	<i>4</i>	<i>5,0697</i>
	<i>Pipe Dy 50</i>	<i>34</i>	<i>169,5397</i>
	<i>Pipe Dy 150</i>	<i>22</i>	<i>957,4884</i>
	<i>Pump</i>	<i>2</i>	<i>8598,9696</i>
	<i>T-branch Dy 150</i>	<i>4</i>	<i>21,0098</i>

Fig. 5.2.23. Result of the report update

Data in the form of links

13. Check the way the links are displayed in the report when objects' property values are modified. For this purpose, go to the source document using the **Open source document** command from the context menu of the report's menu in the document.
14. Modify the component's property value. For example, edit the component's name in the Design Tree and rebuild the model by clicking the **Rebuild (model)** button in the **View** bar. In this example, the *Container* components are renamed into *Vertical tank*.

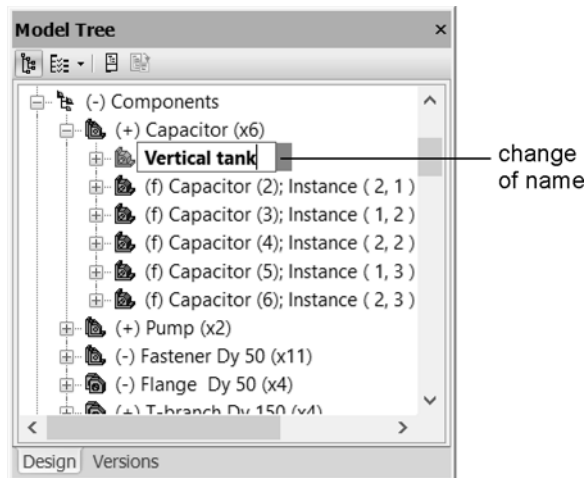


Fig. 5.2.24. Changing property values in the source document

15. Go to the document in which the report is placed. Color of the property values' text has been changed to the color of links requiring the update. At the same time, the report is displayed as crossed. It means that there are modifications in the source document.

<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	Capacitor	6	3901,2439
	Fastener Dy 50	11	70,9003
	Flange Dy 50	4	5,0697
	Pipe Dy 50	34	169,5397
	Pipe Dy 150	22	957,4884
	Pump	2	8598,9696
	T-branch Dy 150	4	21,0098

Fig. 5.2.25. Associative report after modification of property values in the source document

16. Update the report. For this purpose, call the **Update table** command from the context menu of the report. After the update, the new name is displayed in the report in accordance with sorting rules. Text color will be changed to the color assigned to links.

In the example, the *Container* object after renaming to *Vertical tank* has a new position.

<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	<i>Fastener Dy 50</i>	<i>11</i>	<i>70,9003</i>
	<i>Flange Dy 50</i>	<i>4</i>	<i>5,0697</i>
	<i>Pipe Dy 50</i>	<i>34</i>	<i>169,5397</i>
	<i>Pipe Dy 150</i>	<i>22</i>	<i>957,4884</i>
	<i>Pump</i>	<i>2</i>	<i>8598,9696</i>
	<i>T-branch Dy 150</i>	<i>4</i>	<i>21,0098</i>
	<i>Vertical tank</i>	<i>6</i>	<i>3901,2439</i>

Fig. 5.2.26. Result of the report update

17. Check the way the links are displayed when the link to the source document is lost. For this purpose, close the source document and rename it temporarily.
18. Go to the document in which the report is placed. In the document, values of properties will be displayed in a color that means losing the link's source. The report will be displayed as crossed (Fig. 5.2.27).

<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	<i>Fastener Dy 50</i>	<i>11</i>	<i>70,9003</i>
	<i>Flange Dy 50</i>	<i>4</i>	<i>5,0697</i>
	<i>Pipe Dy 50</i>	<i>34</i>	<i>169,5397</i>
	<i>Pipe Dy 150</i>	<i>22</i>	<i>957,4884</i>
	<i>Pump</i>	<i>2</i>	<i>8598,9696</i>
	<i>T-branch Dy 150</i>	<i>4</i>	<i>21,0098</i>
	<i>Vertical tank</i>	<i>6</i>	<i>3901,2439</i>

Fig. 5.2.27. Associative report with a lost link to the source document

19. In order to restore associative link, set the previous name of the document. Text color will be changed to the color assigned to links.

Report editing

20. Edit the report: insert text and split the report in two tables located horizontally. For this purpose, go the Data processing window by double-clicking the frame of the selected report and calling the **Edit in the data processing window** command from its context menu.



First of all perform editing related rebuilding (updating) tables by model, for example, style setup. Then, makes changes in text, insert and move lines, filter lines and so on.

21. Set up the style. For example, set the number of lines in page to be equal to 10. To do so, click the **Setup** button on the Property bar. In the **Style parameters of report** dialog, click the **Layout** button and enter the value.



22. Update the report in the Data processing window by clicking the **Rebuild table** button in the **View** bar.



23. Specify text of the *Equipment* section (Fig. 5.2.28). For this purpose, insert three empty rows in any place of the table using the **Insert rows below** command in the **Report's table bar**.
24. Select the inserted rows and move them up in the table using the **Move row up** command.
25. Enter the text edit mode by double-clicking an empty row and enter simple *Equipment* text. Apply underlining to it.
26. In the same way, specify text of the *Pipes and fittings*.
27. Edit the *Pump* row by adding simple text after the link.

<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	<u><i>Equipment</i></u>		
	<i>Centrifugal Pump</i>	2	8598,9696
	<i>Vertical tank</i>	6	3901,2439
	<u><i>Pipes and armature</i></u>		
	<i>Fastener Dy 50</i>	11	70,9003
	<i>Flange Dy 50</i>	4	5,0697
	<i>Pipe Dy 50</i>	34	169,5397
	<i>Pipe Dy 150</i>	22	957,4884
	<i>T-branch Dy 150</i>	4	21,0098

Fig. 5.2.28. Report editing in the Data Processing Window.



28. Divide the table in two parts. For this purpose, select the row from which the second table should begin. Then, click **From new page** button in the **Report's table bar**.
29. To position tables horizontally, choose the **Down, then to the right** option in the **Assembly** group of the Property bar.
30. Confirm editing by clicking the **Create report** button on the Special Control Panel.



<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>	<i>Designation</i>	<i>Name</i>	<i>Number</i>	<i>Mass</i>
	<u><i>Equipment</i></u>				<u><i>Pipes and armature</i></u>		
	<i>Centrifugal Pump</i>	2	8598,9696		<i>Fastener Dy 50</i>	11	70,9003
	<i>Vertical tank</i>	6	3901,2439		<i>Flange Dy 50</i>	4	5,0697
					<i>Pipe Dy 50</i>	34	169,5397
					<i>Pipe Dy 150</i>	22	957,4884
					<i>T-branch Dy 150</i>	4	21,0098

Fig. 5.2.29. Edited report in a document

The report's text can be edited not only in the Data processing window but also in the fragment's field. For this purpose, double-click the report's line. Then, modify text of the cell you need.

Insertion of names

31. Insert names of each table. For this purpose, call the **Name...** command from the context menu of the report.
32. Enter texts in the dialog fields.

In the example, the following texts are inserted: in the first field — *Test site*, in the second field — *Table continued*.

Test place				Continue table			
Designation	Name	Number	Mass	Designation	Name	Number	Mass
	Equipment				Pipes and armature		
	Centrifugal Pump	2	8598,9696		Fastener Dy 50	11	70,9003
	Vertical tank	6	3901,2439		Flange Dy 50	4	5,0697
					Pipe Dy 50	34	169,5397
					Pipe Dy 150	22	957,4884
					T-branch Dy 150	4	21,0098

Fig. 5.2.30. Report with names of tables

Exploding a link



33. Remove the associative link to the source document from the report, if you do not need it any longer. For this purpose, call the **Explode** command from the context menu of the report. The frame as a sign of associativeness is not displayed.

34. Validate links as was described above (pp 13- 15).



35. In order to rebuild lines, click the **Rebuild** button in the **View** bar. In order to work with reports in fragments, it should be added to the **View** bar, since it is not available by default.

6. Working with BOMs

6.1. General BOM Data

6.1.1. BOM Item

BOM item is a line or a number of successive lines of KOMPAS-3D BOM related to the same material item.

6.1.1.1. BOM Item and Its Properties

A BOM item is of information nature. It represents a set of diverse data on some material item (e.g. part, assembly or document) included in the BOM.

Applying this terminology to paper BOM, we can say that its item is a line (or a number of lines) containing data on one and the same subject (Figure 6.1.1).

Format	Zone	Ref. No	Designation	Name	Qty.	Note
				<u>Documentation</u>		
A2			AEKT.620840.200CB	Assembly drawing		← BOM object
A4			AEKT.620840.200TC	Passport		← BOM object
				<u>Assembly units</u>		
A3	1		AEKT.620840.210	Level Indicator	1	← BOM object
A4	2		AEKT.620840.220	Pump	1	← BOM object
				<u>Parts</u>		
A2	6		AEKT.620840.201	Case	1	← BOM object
A3	7		AEKT.620840.202	Cover	1	← BOM object
A4	8		AEKT.620840.203	Plug	2	← BOM object

Fig. 6.1.1. BOM items

Comparing a BOM to a database, we can say that the BOM item is similar to database record. The BOM items are classified as **base** and **auxiliary** items.

6.1.1.1.1. Base Item

The base items have features of columns autocomplete, sorting, attaching graphical items from assembly drawing, attaching parts from assembly, etc.



It is recommended that the base items be created wherever possible in the process of BOM completion.

6.1.1.1.2. Auxiliary Item

Unlike the base item, no service functions, which are supported by BOM, are provided for auxiliary item. The automatic sorting of auxiliary items is not supported, etc.

It is recommended that the auxiliary items be used to achieve the BOM formatting, which cannot be achieved by introducing base items (Figure 6.1.2). For example, by using auxiliary BOM item, desired text (comment) can be entered in the BOM table, or an empty line can be created in the middle of a section (for more details see Section 6.2.2.3.1).

Format	Zone	Ref. No	Designation	Name	Qty.	Note
				<u>Standardized Products</u>		
				.Screws GOST R 11738-84		← Auxiliary object
						← Auxiliary object
		14		2 M6 x 1,25-6g	12	← Main Item
		15		2 M8 x 1,25-6g	4	← Main Item
		16		2 M10 x 1,25-6g	6	← Main Item
		17		2 M12 x 1,25-6g	6	← Main Item

Fig. 6.1.2. Base and auxiliary BOM items

6.1.1.2. Structure of the BOM Item

External representation of a BOM item is its text placed in the lines of a BOM form. This text corresponds to a line in the “paper” BOM.

However, in contrast to such line, the item of the electronic BOM contains much more information.

As a rule, the base BOM item consists of

- text portion,
- geometry (graphical items and 3D models),
- set of additional parameters.

The auxiliary BOM item does not contain geometry and has less auxiliary parameters than a base item.

6.1.1.2.1. Text Portion of the BOM Item

The text portion of the BOM item contains data that is entered in the lines of the BOM table. For example, objects illustrated in Fig. 6.1.1, have the following text portions “A2 AEKT 620840.200SB Assembly Drawing”, “A4 AEKT 620840.200PS Passport”, “A3 1 AEKT 620840.210 Level Indicator 1”, “A4 2 AEKT 620840.220 Pump 1”, “A2 4 AEKT 620840.201 Casing 1”, etc.

Text entry and editing methods do not differ practically from those used in operations with drawings. User has access to commands for text and paragraph formatting, insertion of fractions, indices, super- and subscript, special characters, symbols and predetermined text.

Besides, **the text templates** are used for creation of some items, in which the text is generated on the basis of standard rules. Text Templates is a specific mechanism used only in operations with BOM items.

More details about the text entry and editing methods for BOM item are provided in Section 6.2.2.4.

In displaying text of BOM items, the contents of the BOM title block is taken into consideration. If the contents of cells *Designation* (disregarding the code) and *Name* (disregarding title of document) of the item fully matches the contents of the relevant cells of the title block, the part name display in the text of the BOM item will be automatically disabled. The main purpose of this mechanism is to provide correct presentation of the BOM items in the *Documentation* section (for more detail see Section 6.2.2.10).

6.1.1.2.2. Geometry of the BOM Item

The BOM item may contain information on

- the assembly drawing graphical items, of which the item's image is made up, and (or)
- the relevant component of a 3D assembly.

This information is called **geometry of the BOM item**.

For instance, during creation of the BOM item "*Shaft*" the line sections, hatches and other elements, which depict the shaft on assembly drawing, as well as the part *Shaft* from 3D assembly, were specified as its geometrical components.

The BOM item cannot include geometry from any document (drawing or assembly), but only geometry from the document attached to this BOM (see Sections 6.1.3.5 and 6.2.4.1).

The BOM supports viewing the geometry of BOM items (see Section 6.2.2.5.3). In this mode, the system automatically selects the geometry related to the selected BOM item in a drawing or assembly (provided that the drawing or assembly are open). When moving through the BOM (e.g. by pressing the arrow keys), the images of the BOM items are highlighted in the neighboring window.

This mode is rather helpful for handling the assemblies containing scores of components. It allows to locate quickly the image of any BOM item (assembly unit, part, standard part, etc.) in the assembly drawing or in the model.



It is not mandatory to include all graphic items, which make up the image of an item, in the item geometry. However, it is highly recommended to include the reference leader in the item geometry.

Thanks to the inclusion of the reference leader in the graphical composition of the BOM item, some functions of associative link between the BOM and the assembly drawing are implemented, namely ensuring that the reference numbers on the drawing and those in the BOM are matching, as well as automatic calculation of zones.

- After you have changed some reference numbers in the BOM, it is sufficient to transfer these changes to the drawing (see Sections 6.1.3.6.2 and 6.2.4.2.2), in which new reference numbers will appear automatically on the sides of the leaders.

- If zone calculation (see Section 6.3.2.1) is included in the BOM, new designations of zones, in which every reference number is located (see Sections 6.1.3.6.1 and 6.2.4.2.1), may be automatically transferred to the BOM after changing the reference numbers of the leaders in the drawing (e.g. due to zoom in/out or due to changed format of the drawing)

The BOM settings may be changed, so that after deleting a BOM item its relevant geometry is automatically deleted, and after deleting the geometry the BOM item is automatically deleted (see Section 6.3.2.1). For instance, if the “Gasket” item is removed from the BOM, the image of the gasket in the drawing also disappears, and vice versa.



This feature of the BOM is to be used with care, because it is not always possible to restore a mistakenly removed geometry.

6.1.1.2.3. Additional Parameters of the BOM Item

The BOM item may contain a lot of diverse information. It supplements the text portion of an item that is visible in the BOM line.

This information is never visible on the BOM form. However, it may be viewed and edited at any time.

All types of this information are called **additional parameters** of the BOM item.

The additional parameters of the BOM item include

- the settings of this item,
- documents attached to the item,
- data in additional columns of the item.

Settings of the Item

The settings of the item determine its position and display in the BOM, as well as its behavior during the automatic assignment of reference numbers and sorting of items (see Section 6.2.2.6.1).

Attached Documents

Documents of the KOMPAS-3D system may be attached to the BOM item (see Section 6.2.2.6.3). This is a two-way associative link. It allows

- automatic data exchange between the text portion of the item and the attached documents (see Section 6.2.4.2.2),
- automatic creation of a draft drawing of a part based on its image in assembly drawing (see Section 6.2.6.4), as well as facilitates access to documents.

Additional Columns

A BOM item may contain data supplementing the information included in the standard form. This data is stored in so-called “additional columns” of the BOM item (see Section 6.2.2.6.2). They may be viewed or edited at any time. However, they are not visible in the BOM form and are not printed out.

An example of information in additional columns are item's weight and cost. Any other item-related information may be also included in additional columns (OKP Code, material, text com-

ments, etc.). The number and contents of the additional columns are determined by user's demands.

The data in the additional columns may be summed automatically (see Section 6.2.6.2). Usually this feature is used for cost estimation of the item components and calculation of the weight of the item (when 3-D model is unavailable).

6.1.2. BOM Structure

The BOM item is a basic structural unit of the BOM.

Just as a drawing consists of geometrical items and macro elements, and text documents consist of words and sentences, the BOM consists of BOM items grouped in sections.

The items in the BOM table interlace with **titles of sections**, **titles of blocks**, **empty lines** and **reserved lines**.

6.1.2.1. Sections

In any competently compiled BOM, the items are arranged in a sequence required by the standard. As a rule, the standard specifies certain sections which should be included in BOM.

Sections are one of the components of the BOM style (see Section 6.3.1). Their number, titles and sequence are formed in style settings. By selecting the style of a current BOM, you, in addition to other settings and parameters, select a list of possible BOM sections.

During item creation, you may select the section of the location of the item.

It is not possible to move an item from one section to another.

Sections without items cannot exist in BOM. Therefore, when a new section is created, the first item is immediately generated in it, and when the last item is deleted from a section the section itself is also deleted.

Section deletion is only possible by sequential deletion of all items in the section.

Current section is the section, in which one of the items is selected or edited.

A particular feature of the BOM of the system KOMPAS-3D is the possibility to create and fill in sections in arbitrary sequence. You may start with adding standard parts, then create and fill out the *Documentation* section, then proceed with adding parts and after that add assembly units.

The system will automatically arrange the created sections in a standard sequence (in the general case, the sequence assigned in the BOM style settings will be used). Every new section will be placed in a fixed location in relation to the other sections, moving apart the filled-out line, as required.

6.1.2.1.1. Section Blocks

The BOM may contain embedded and additional sections.

Embedded sections are placed inside a section, after all items of this section.

Additional sections are placed at the end of the BOM, after all its sections. Normally, the additional sections are placed on a new sheet.

The groups of embedded and additional sections are called **blocks of embedded sections** and **blocks of additional sections**, accordingly.

6.1.2.1.2. Titles of Sections and Blocks of Sections

As a rule, the title of every section or block of sections in a BOM is placed in a separate line (or in a number of lines) in the beginning of the section or block. These lines are called **titles of sections** and **titles of blocks**, accordingly. E.g. Figure 6.1.3 shows titles of sections.

The BOM settings may be selected so that the section titles (including the additional and embedded sections) are not displayed in the BOM form (see Section 6.3.2.2). The same setting is available for titles of blocks (see Sections 6.3.2.2.3 and 6.3.2.4).

6.1.2.2. Empty Lines

An **empty line** is a line in the BOM form, which is placed immediately above or under the title of a section or a block of sections. It separates the title from the BOM items (Figure 6.1.3).

No text can be entered in the empty line. Its presence in the BOM is determined by the standard.

The display of empty lines around the section titles can be disabled when setting up the sections (see Section 6.3.2.2), and the empty lines around the block titles can be disabled when setting up the blocks (see Sections 6.3.2.2.3 and 6.3.2.4).

6.1.2.3. Reserved Lines

Reserved line is a BOM line for the changes, which will be later added to the BOM issued (printed on paper) (Figure 6.1.3).

The presence of reserved lines in the BOM is determined by the standard.

The system automatically creates a number of reserved lines (their number is specified by the user) in each section.

These lines are always placed at the end of a section (whatever sequence was used to add items of BOM) and no data can be entered in those lines.

The placement of reference numbers in each section takes into account the number of reserved lines in the previous section (see Sections 6.1.2.6 and 6.2.2.8).

	Format	Zone	Ref. No.	Designation	Name	Qty.	Note
Section							← Empty line
					<u>Documentation</u>		← Section header
							← Empty line
A2				AEKT.620840.200CB	Assembly drawing		
	A4			AEKT.620840.200TC	Passport		
Section							← Reserved text line
					<u>Assembly units</u>		← Section header
							← Empty line
A3	1		AEKT.620840.210	Level Indicator	1		
A4	2		AEKT.620840.220	Pump	1		
Section							← Reserved text line
					<u>Parts</u>		← Section header
							← Empty line
A2	4		AEKT.620840.201	Case	1		
A3	5		AEKT.620840.202	Cover	1		
A4	6		AEKT.620840.203	Plug	2		
							← Reserved text line

Fig. 6.1.3. Titles of sections, empty and reserved lines

6.1.2.4. Blocks of Variants

The standard stipulates a number of forms for group BOM. Maximum 10 columns are provided in these forms for adding a number of variants.

One of the ways to fill out a BOM for a product existing in more than 10 variants is to divide them into groups containing 10 variants each and put them in the same BOM by first entering data about variants from the basic variant to the 9th variant, then from the 10th to the 19th variant, etc. These groups are called **blocks of variants**.

If a block of variants begins in the top of the sheet, the variant numbers are placed in the BOM headline. If a block of variants begins in the middle of the sheet, the blocks are separated from each other by the line “*Designation of variants*” containing in the columns *Number of variants* the designation of the variants of the next block. This line (Figure 6.1.4) is called **beginning of the block**. It, as well as the section title, is framed with empty lines on top and bottom.

Formal Zone Ref. Ac	Designation	Name	Qty. of variants									Note		
			-	1	2	3	4	5	6	7	8		9	
														← Empty line
		<u>Designation of variants</u>												← Beginning of the block
														← Empty line
		<u>Assembly units</u>												← Section header
														← Empty line
Block of variants	Section	A1 1	ADPK 137563.100	Gear box	1	1								
		A1 1	ADPK 137563.100-01	Gear box			1		1		1		1	
		A1 1	ADPK 137563.100-02	Gear box				1		1		1		1
														← Reserved text line
														← Empty line
		<u>Parts</u>												← Section header
														← Empty line
Block of variants	Section	I2 3	ADPK 137563.009	Bonnet	1	1		1		1		1		
		I2 4	ADPK 137563.012	Clamp		2	2		2		2		2	
														← Reserved text line
														← Empty line
		<u>Designation of variants</u>												← Beginning of the block
														← Empty line
		<u>Assembly units</u>												← Section header
														← Empty line
Block of variants	Section	A1 1	ADPK 137563.100-01	Gear box	1		1		1		1			
		A1 1	ADPK 137563.100-02	Gear box		1		1		1		1		
														← Empty line
		<u>Parts</u>												← Section header
														← Empty line
Block of variants	Section	I2 3	ADPK 137563.009	Bonnet	1	1		1		1				
		I2 4	ADPK 137563.012	Clamp		2	2		2		2			
														← Reserved text line

Fig. 6.1.4. Blocks of variants

The display of lines containing beginnings of blocks and empty lines around them may be disabled through the settings of the blocks of variants (see Section 6.3.2.3).

6.1.2.5. Item Sorting

The BOM provides a feature of automatic sorting of items inside sections.

The rules of sorting are a BOM style component (see Section 6.3.1).

The rules of sorting of the items (sorting type and column for sorting) are established for every section. The BOM items are sorted by one column in the section (different columns for sorting may be used in different sections).

The auto-sorting of items in every section can be enabled/disabled in the settings of the current BOM (see Sections 6.2.2.7 and 6.3.2.2), however the rules of sorting cannot be changed. The rules of sorting for the current BOM may be changed by replacing the BOM style with the style containing the desired rules of sorting.

The user can disable the auto-sorting of items for any section. All items of the section, in which auto-sorting is disabled, can be arranged in arbitrary sequence.

If auto-sorting is enabled for the section, it takes place every time a new item is created in the section and after editing the current item's text.

The auto-sorting in the sections, which contain subsections, is carried out inside the subsections. The sequence of subsections is left untouched and the items are not moved between subsections.

There are five types of item sorting:

- Ascending,
- Descending,
- Complex ascending,
- Complex descending,
- For document section.

6.1.2.5.1. Ascending Sorting and Descending Sorting

The **ascending** sorting may be performed in a different way depending on the data type in a column.

In the columns with numerical type of data, items are sorted by comparing their values. As a result of the sorting the numbers are arranged in ascending order.

Example of list of numerical values after ascending sorting:

25

125

250

In columns with the “line” type of data, the ascending sorting is done by way of comparing symbols in the relevant positions of the lines (starting from position 1). As a result of this sorting, the line, in which the first differing symbol is closer to the beginning of the alphabet, will be placed first. Digits are placed before letters and Latin letters are placed before Cyrillic letters. An example of ascending sorting is the word placement in most dictionaries and encyclopedias.

An example of list of designations after ascending sorting:

A125GP

A25MP

D250MP

D25GP

In this case, the initial symbols of lines were compared first. The lines beginning with *A*, were placed before the lines beginnings with *D*. Inside the first group of lines (*A25MP* and *A125GP*) the second symbols were compared, *1* and *2*. The line containing *1* in the second position was placed first. The further comparison of lines beginning with *D* showed that their second and third symbols were identical. As a result of the comparison of the fourth symbols (digit *0* and letter *G*), the line containing a letter was put in the last position.

The **Descending sorting** depends on the type of data in the column. The rules of descending sorting are reverse to those of ascending sorting.

The lists in the previous example, which were sorted in ascending order, will look as follows after descending sorting:

250

125
25
and
D25GP
D250MP
A25MP
A125GP

6.1.2.5.2. Combined Ascending/Descending Sorting

The rules of the **combined** sorting combine the rules of sorting for columns with different types of values. In this case, the lines are broken down into letters and numbers, which are composed of a sequence of digits. After that, letters and numbers are compared sequentially. The main difference between combined sorting and descending/ascending sorting is that the numbers located in any place of the line are taken into account. Whereas during ordinary sorting the digits are compared against digits and the letters against digits, in the combined sorting the numbers are compared against numbers.

The lines of the previous examples after **combined ascending sorting** are placed in the following order:

A25MP
A125GP
D25GP
D250MP

At first, the first symbols were compared. As a result, the strings starting with letter *A* took the first positions in the list. During further sorting of the *A25MP* and *A125GP* strings, not the digits *2* and *1*, but the numbers *25* and *125* were compared. As a result, the line containing the smallest number *25* was put in the first position. In the lines *D25GP* and *D250MP*, the numbers *25* and *250* were compared, rather than the symbols *2* and *2*, *5* and *5*, *G* and *O*. As a result, the line containing the biggest number *250* was put in the last position.

The same list after **combined descending sorting** (whose rules are opposite to those of combined ascending sorting) looks as follows:

D250MP
D25GP
A125GP
A25MP

6.1.2.5.3. Special Cases of Combined Sorting

A more complicated manifestation of combined sorting is the sorting of texts created from text templates (usually represented by names of standard products and materials, for more details see Section 6.2.2.4.5). The texts created on the basis of templates consist of several fields. These texts often require sorting according to certain sequence of fields based on values in those fields, but not according to the sequence of symbols and numbers they consist of. For instance, standard products are first sorted by name, then by standard designation, then in as-

ending order of basic parameters and sizes. Example of combined sorting of standard products:

*Key 2–10*8*40 GOST 23360–78*

*Key 2–14*9*50 GOST 23360–78*

*Stud 10*45 GOST 3128–70*

*Stud 12*50 GOST 3128–70*

The information about the reference fields for sorting and the sorting sequence for the given type of items is stored in a text template. More details about text templates are provided in Section 6.3.5.

In some cases, neither ascending nor descending sorting is suitable for the correct layout of the generated texts based on templates. In these cases, the sorting sequence for values in the fields is set clearly in template settings, by listing possible values and arranging them in required sequence.

For instance, designations of electrical elements of the same name are sorted first in ascending order of the measurement units and then in ascending order of the nominal value:

Resistor MLT-0.25-210 Ohm SP 11–85

Resistor MLT-0.25-240 Ohm SP 11–85

Resistor MLT-0.2524 KOhm SP 11–85

Resistor MLT-0.25-27 KOhm SP 11–85

The BOM columns, in which the text is generated and sorted on the basis of templates, have the value type of “record”.

6.1.2.5.4. Sorting for a Documentation Section

The sorting **for a documentation section** is enabled, as a rule, during the setting-up of the *Documentation* section. In this type of sorting the last two symbols of line are taken into account: the system interprets them as the document reference number. First the lines are sorted alphabetically without the two last symbols. Inside the resultant groups with the same beginning of lines, the lines are sorted by the last two letters. Sequence of documents is observed according to GOST 2.102–68.

Example of sorting document designations in the *Documentation* section:

318.606100.00 SB

318.606100.00 KZ

318.606100.00 PZ

318.606100.00 RR

The listed documents are placed in the following order: assembly drawing, general kinematic diagram, explanatory note, calculations.

6.1.2.6. Creating Reference Numbers

From designer’s viewpoint, the main connecting link between an assembly and the respective BOM are the reference numbers. Only thanks to these numbers on the leader borders and in the column *Position*, it is possible to establish correlation between the subject (part, unit) on the drawing and the BOM line.

Such correlation can be automatically supported by the BOM of the system KOMPAS-3D. The position number of the BOM item is transferred to the leader border on the drawing.

The position numbers can be also automatically assigned to BOM items. During the creation of base BOM items, the numbers reflecting sequence of items creation will appear in their columns *Position*.

The user can disable the placement of the positions in any section (see Section 6.2.2.8).

Any base item can be adjusted, so that its position is omitted in the BOM table, even if the item is located in the section in which the placement of positions is enabled (see Section 6.2.2.6.1).

As a result of items sorting or creation in a way different than the sections' sequence order, the items numbering may differ from their order of placement in the table (Figure 6.1.5, a). To eliminate this non-conformance, the command **Service — Arrange Positions** is used, which calculates and automatically arranges the positions of the BOM items (Figure 6.1.5, b).

Format	Zone	Ref. No	Designation	Name	Qty.	Note
				<u>Assembly units</u>		
A3		1	AEKT 620840.210	Level Indicator	1	
A4		2	AEKT 620840.220	Pump	1	
				<u>Parts</u>		
A2		4	AEKT 620840.201	Case	1	
A3		5	AEKT 620840.202	Cover	1	
A4		6	AEKT 620840.203	Plug	2	
A4		7	AEKT 620840.204	Left latch	1	
A4		8	AEKT 620840.205	Right latch	1	
A4		9	AEKT 620840.206	Grid	3	
A4		10	AEKT 620840.206-05	Grid	3	
A4		11	AEKT 620840.206-06	Grid	3	

a)

Format	Zone	Ref. No	Designation	Name	Qty.	Note
				<u>Assembly units</u>		
A3		1	AEKT 620840.210	Level Indicator	1	
A4		2	AEKT 620840.220	Pump	1	
				<u>Parts</u>		
A2		4	AEKT 620840.201	Case	1	
A3		5	AEKT 620840.202	Cover	1	
A4		6	AEKT 620840.203	Plug	2	
A4		7	AEKT 620840.204	Left latch	1	
A4		8	AEKT 620840.205	Right latch	1	
A4		9	AEKT 620840.206	Grid	3	
A4		10	AEKT 620840.206-05	Grid	3	
A4		11	AEKT 620840.206-06	Grid	3	

b)

Fig. 6.1.5. Numbers in the **Position** column: a) those, which were generated during the addition of items; b) those that were arranged using the **Arrange Positions** command

The arrangement of the positions takes into account the sequential order of the items in the BOM, the number of reserved lines in each section, the disablement of the position placement in some sections, as well as the settings of the items. A reference number is assigned to every base item according to its individual settings and its location in the BOM. It does not refer to the items in the sections, in which the position placement function is disabled.

If the position placement is disabled in the section, numbers do not appear in its column *Position* during adding of items or during the automatic position placement. Position numbers, which follow the position numbers of previously numbered items, are assigned to the items of the next section (provided that the position placement is enabled).

The positions are placed in the sections (if the function is enabled) even if the displaying of the positions of all or some items of the section is disabled. It may happen that the position numbers in the section are not visible, but they are taken into account in the numbering of items in the next sections.

In some cases, several successive BOM items may have the same position numbers (see Section 6.2.2.6.1). At that, displaying of the repeated numbers can be disabled. The part variants in a group BOM are an example of items having the same position numbers.

The transfer of position numbers from the BOM to the leader borders in a drawing is only possible for those BOM items which include a reference leader (see Section 6.2.2.5). This transfer is carried out during data synchronization (see Section 6.2.4.2.2).

6.1.3. Interfaces between the BOM and other documents

Drawings, fragments, parts and assemblies may contain the same BOM items as the BOM document.

By default, these items are not visible in graphical documents and in the models in construction mode and are not printed out¹. However, you may always view and edit them.

The special slave mode (see Sections 6.1.3.3 and 6.2.3.5) is intended for viewing and editing the BOM items contained in drawings, fragments, parts and assemblies.

6.1.3.1. BOM Items in the Drawings

Usually images of parts and units, reference leaders and other data in assembly drawings are added prior to the creation of a separate BOM document.

With the traditional (paper-based) BOM creation method, the designer remembers all this information or records it in a draft document prior to filling in a BOM. The use of the system KOMPAS-3D allows storage of the data required for BOM creation based on the current drawing, in the drawing itself.

The BOM items should be created in the drawing as their images are constructed (see Section 6.2.3.1). After the BOM has been attached to the drawing (see Section 6.2.4.1.1), these items can be transferred to it whenever necessary (see Section 6.2.4.2.1). The items are not removed from the drawing, but simply copied to the BOM (together with all their additional parameters and the information on the attached geometry).

The BOM items can be edited both in the BOM document and in the drawing (in slave mode). After the BOM item has been edited in one of the attached documents (BOM or drawing), the changes can be transferred to another document through synchronization (see Section 6.2.4.2).

If an assembly drawing consists of several sheets (i.e. from several KOMPAS-3D documents), then BOM items can be created in each of them and transferred to the BOM attached to all the sheets. The items from all attached drawings will be arranged in the BOM according to the order stipulated in the standard (each of them will be placed in the section, in which it was created in the drawing, and all items will be sorted).

Since the transfer of BOM items from the drawing to the BOM requires only one command to be called and takes little time (normally some seconds), it can be used for quick restoration of

1. In the drawing you can enable the mode under which the BOM table with the items located in it will be placed above the title block and printed out together with the drawing (see Section 6.2.6.12).

a damaged or lost BOM. If you are going to create items in a separate BOM document without further transfer to the drawing, such restoration will be impossible.

Another advantage of the creation of items in a drawing is the data acquisition by an item directly from the drawing. If you create a BOM item containing a reference leader in its geometrical structure in the drawing, the column *Zone* will automatically display the designation of the drawing zone in which the attached geometry is located, and after every change of the position number of the BOM item (e.g. after sorting) a new number will appear on the border of the relevant leader in the drawing.

6.1.3.2. BOM Items in the Models

The BOM items can be created not only on a “flat” assembly drawing, but also in a 3D model (in a part or assembly). BOM items can be formed in the model at any stage of its creation (see Sections 6.2.3.3 and 6.2.3.4).

The BOM items can be edited both in the BOM document and in the model (see Section 6.2.3.5). After a BOM item has been edited in one of the attached documents (in a BOM or model), the changes can be transferred to another document through synchronization (see Section 6.2.4.2).

6.1.3.2.1. Part

A part may have an unlimited number of BOM items, which belong to any section of BOM. In most cases however, a single BOM item containing the BOM's name and designation and belonging to Section *Parts* is created in the part.

When parts are added to the assembly, the BOM items of this part go into this assembly, and from there they are transferred to the sections of the BOM in which they were created.

6.1.3.2.2. Assembly

There may be two types of BOM items in an assembly: external and internal.

External Items of a BOM

External items of BOM in the assembly are the items of BOM intended to be transferred to the assemblies in which the given assembly will be added as a subassembly.

External BOM items are generated by user (see Section 6.2.3.4.1). They may belong to any sections of the BOM. In most cases however, a single BOM item containing the BOM's name and designation and belonging to section *Assembly units* is created in the assembly.

When the assembly is inserted into some other assembly, the external BOM items of this assembly go into the main assembly and from there are transferred to the BOM sections, in which they were created.

Internal BOM Items

Internal BOM items in an assembly are the items which will not be transferred to another assembly when the given assembly is inserted into it as a subassembly.

They can be generated by the user, or “come” to the assembly from other models during the insertion of components (parts and subassemblies) into the assembly, its internal BOM items

are created automatically. The external BOM items belonging to subassemblies and the BOM items belonging to parts are transferred to the assembly.

The internal items are transferred to the BOM document which is attached to the assembly (see Section 6.2.4.2.1). The items are not removed from the assembly, but are copied to the BOM (together with their additional parameters and the information on the attached geometry).

The internal BOM items are also transferred to the drawings during the creation of associative views containing images of models. Thanks to the above, it is sufficient to create each BOM item only once— in a model, when preparing a complete set of associative documents (models, drawing and BOM). The item is transferred from this model to higher-level models, drawings and BOM.

The user can supplement a set of internal BOM items by creating new items, which are not associated with external files (see Section 6.2.3.4.2). If required, these internal items can be connected with one or several bodies created in the same assembly (see Section 6.2.2.5.1).

The user can also edit internal items generated automatically by the system.

As required, the user may exclude any BOM item belonging to a part (subassembly) from an assembly or a drawing.

6.1.3.3. Slave Mode

The **slave mode** is a viewing and editing mode allowing to edit and view BOM models directly in a graphical document or in a model.



This mode is available if a BOM description exists in the document. The slave mode is initiated by **BOM — Edit items** command.

A special window with columns and sections similar to the columns and the sections of the BOM is used to work with BOM items in a graphical document or a model. In practical terms, this window does not differ greatly from the edit box of a BOM document. The only difference is that the title shows the name of the document, in which the BOM items are located, and the comment “► *BOM items*”, rather than the name of the BOM document.

In slave mode, all operations with BOM items are accessible. The only exception is that it is impossible to call the position numbers placement command.

The items, which have been created and edited in slave mode, are permanently stored in a graphical document or in a model. Any time they can be transferred to the BOM, which is attached to the document.

Appearance of the mode symbol in the document window indicates that the BOM is in the subordinate mode. In order to exit the mode, click the symbol.

6.1.3.4. Description of BOM

The description of a BOM incorporates the file name of the BOM attached to a graphical document, the style of this BOM, and display settings for the mass values.

If a model or a drawing does not contain any BOM item, there is no description of BOM in this document. When the first BOM item is created in a document (model or drawing), this document is provided with description of BOM. An item is created according to the style selected in the system for new BOM documents. Therefore, the generated description contains the style

of the BOM, which is selected in the process of setting up new documents. The setting of displaying the mass values also conforms with the default one for new BOMs. The description will not have a BOM file name until a BOM document is attached to the model or the drawing.

If a model or a drawing is attached to a BOM during manipulations with the BOM (see Section 6.2.4.1), the attached document will be provided with a description. The description incorporates the style and the BOM file name. Once the attachment has been made, the items in the drawing will be created according to the style of the attached BOM. This is done to ensure that the items in the drawing and the attached BOM have the same structure (columns, rules of sorting), and the appearance of the documents is kept unchanged during the exchange of items between documents.

The BOM description for a fragment contains only the BOM style and a setting of displaying the mass values. This is due to the fact that the fragments are not directly attached to the BOM, however it is possible to create BOM items in a fragment for further transfer to drawings (see Section 6.2.3.1.2).

Usually a model or a drawing contains one description of BOM, which allows the creation of items for further transfer to a single BOM. Nevertheless, it is permitted to create several descriptions of BOM in the same document. This is detailed in Section 6.2.6.14.

If a model or a drawing have several descriptions of BOM, there is only one current description at a given moment. The BOM items are created according to the style included in the current description of BOM. The user can at any time make current some other description of BOM.

Work with the BOM descriptions present in the document is performed using the dialog created by the **BOM – BOM description control...** command. Here, you can create a BOM description, edit it, make it current or delete it.

6.1.3.5. Links of Documents with the BOM

As a rule, there are associative links inside a set of documents of a product, which are created in KOMPAS-3D. The data exchange between documents takes place thanks to these links.

6.1.3.5.1. Relation Between an Assembly Drawing and a BOM

According to the design practices, a BOM of a product always matches the assembly drawing of this product.

A KOMPAS-3D BOM may be also linked with an assembly drawing (one or several KOMPAS-3D drawings). This is a two-way associative link.

The user can specify the assembly drawing, with which the current BOM correlates (see Section 6.2.4.1.2). As a result, the drawing receives and stores information about the BOM it is attached to. This link may be established from the current assembly drawing by specifying the BOM which has been developed for it (see Section 6.2.4.1.1). So, the information on a drawing attachment will be transmitted to the BOM.

Thus, after a link has been established between the BOM and the assembly drawing, the information about this link is stored in both documents, and the attachment method (establishing attachment from the drawing or from the BOM) will be exclusively the user's choice.

With the BOM window active, you can quickly open the attached drawings. And vice versa, during work with a drawing, you can quickly open the attached BOM. Thanks to a once establish

link between the BOM and the assembly drawing, the system remembers the location of the attached documents, and once the command for opening has been transmitted the system finds them by itself.

Another manifestation of the two-way link is the possibility for data transmission from a drawing to the BOM or from the BOM to a drawing, and moreover, such transmission is an associative one (see Section 6.2.4.2).

You can transmit from a drawing to a BOM the designations of zones, in which images of BOM items are located, as well as names and designations of these items. You may transmit from a BOM to drawings the position numbers, names and designations of items.

For example, as a result of the changed scale of the drawing or changed image location, some position numbers were moved to other zones of the drawing. For manual correction of this situation, you will have to scroll all reference leaders, check their position in the zones and correct the designations of the zones in the BOM. The KOMPAS-3D BOM carries out all these actions automatically (by user's command). The designations of the zones, in which the position numbers are currently located, are transmitted to the BOM and placed in its column *Zone*, in the line which has a relevant position number. If the position numbers have been changed in the BOM due to the sorting of items, the position numbers of the corresponding position leader borders in the drawings will also change.

So, the changes made in one document are transmitted to an unambiguously specified relevant location in the other document.

6.1.3.5.2. Link Between an Assembly and the BOM

In addition to a link with an assembly drawing, the KOMPAS-3D BOM can be also linked to a 3D model of assembly. This is also a two-way associative link.

The user can specify the model with which the current BOM correlates (see Section 6.2.4.1.2). As a result the model receives and stores information about the BOM it is attached to. This link may be established from the current assembly by specifying the BOM which has been developed for it (see Section 6.2.4.1.1). So, the information on a model attachment will be transmitted to the BOM.

Thus, after a link has been established between the BOM and an assembly, the information about this link is stored in both documents, and the attachment method (establishing attachment from the assembly or from BOM) is exclusively the user's choice.

With the BOM window active, you can quickly open the attached assembly. And vice versa, during work with a model, you can quickly open the attached BOM. Thanks to a once establish link between the BOM and an assembly, the system remembers the location of attached documents, and once the command for opening has been transmitted the system finds them by itself.

Other manifestation of the two-way link is the possibility for data transmission from a model to the BOM or from the BOM to a model, whereby this is an associative transmission (see Section 6.2.4.2).

You can transmit the designations, names and weights of components from a model to the BOM. You can transmit the designations and names of components from the BOM to a model.

For example, as a result of changed dimensions of parts, their weights also changed. The KOM-PAS-3D BOM will receive updated weight values of the assembly components automatically (by user's command).

If an assembly model has more than one version, it should be linked to a group BOM. Apart from component data, the group BOM also receives version data from the assembly: total number of versions, their numbers and composition. Once the assembly is modified, BOM is automatically updated.

6.1.3.5.3. Links of Documents with the BOM Title Block

When attaching a document — an assembly or an assembly drawing — to the BOM (see Section 6.2.4.1.2), the user may specify

- a document from which the designation and the name of the product will be transferred to the title block of the current BOM,
- as well as the documents which will receive the product designation and name from the current BOM.

6.1.3.6. Data Synchronization

The data in an assembly drawing or model can be synchronized thanks to the attachment of these documents to the BOM.

Synchronization is the process of transfer of BOM items from a drawing or an assembly to an attached BOM or from the BOM to attached documents.

6.1.3.6.1. Item Transfer from a Drawing or Assembly to the BOM

The BOM items created in a drawing or in a model are not visible on the drawing or in the model (except for the cases when the BOM is located on the drawing). They can be seen only in the slave-mode window and cannot be printed out.

To make the BOM items appear in a separate BOM document (which can be printed), they should be transferred from a drawing or from a model to the attached BOM (see Section 6.2.4.2.1).

The synchronization is performed also automatically (without special command by the user) during saving or closing of a document attached to the BOM.

After calling the synchronization command, the changes made in the BOM items are transferred from the document to the attached BOM. Only divergent data are transmitted during synchronization, without absolute copying of all items from one document to the other.

- If some item is missing from BOM, it will be copied therein.
- If there is a difference between the text portion or any additional parameters of some item in the model/drawing and in the BOM, then the text or additional parameters will be copied from the item in the model/drawing to the relevant item in the BOM.
- If some item, which exists in the BOM, is missing from the model/drawing, this item will not be lost as a result of the synchronization. Thanks to this feature, the items from several attached drawings can be transmitted to the BOM: these items will supplement each other without being eliminated. Moreover, the BOM will remember the source drawing for each BOM

item. If items are to be transmitted from the BOM to the drawings later, each of them will be sent to its original drawing.

The BOM to which the changes are transmitted may not be loaded during synchronization (in this case it will be opened, modified and closed in background mode without being displayed).

Upon completion of the synchronization, the system will generate a message to specify which BOM has been modified.



If the data transfer from model/drawing to BOM is disabled by the user in the BOM settings (see Section 6.3.2.1), the synchronization will not be carried out either by user's instructions or automatically.

6.1.3.6.2. Transfer of Items from the BOM to Drawings and Assemblies

Sometimes the changes made in an attached BOM need to be transmitted to an assembly drawing or a model of assembly. For example, new position numbers were assigned to the items in the BOM after sorting. The items with these position numbers should be transferred to the drawing so that they could appear on the leader borders (see Section 6.2.4.2.2).

The synchronization is also carried out automatically (without special command to be called by the user) during saving or closing the BOM.

After calling the synchronization command, the changes made in BOM items are transferred from the BOM to the attached models/drawings. Only divergent data are transmitted during synchronization, without absolute copying of all items from one document to the other.

- If there is difference between the text portion or additional parameters of some item in the model/drawing and in the BOM, then the text or additional parameters will be copied from the item in the BOM to the relevant item in the model/drawing.
- If some item existing in the BOM is missing from the model/drawing, this item will be transmitted to the model/drawing during synchronization, provided that this item contains the geometry from this document. Items, which do not have geometry and exist only in BOM, are never transmitted to models and drawings. If an item without geometry was received in the BOM from a model/drawing, it will be sent to its original document.

The drawing, to which the changes are transmitted, cannot be loaded during synchronization. In this case, it will be opened, modified and closed in background mode without being displayed.

During synchronization, the changes are also transmitted to the documents attached to the BOM items (as required). For instance, during linking a part to an item of a drawing (see Section 6.2.2.6.3) the option **Transfer Changes to a Document** was enabled, and thereafter the part designation in the BOM was changed. In this case, in the process of synchronization, the new designation will be transmitted not only to the BOM item in the assembly drawing, but also to the relevant column of the title block in the drawing of the part.

After completing the synchronization, the system will generate a message enumerating the documents that have been modified.

6.1.3.6.3. Data transfer between the BOM main text and connected documents

No special command needs to be called by the user to transfer the product designation and name between the title block of the BOM and the attached documents. This transfer is carried out automatically during document saving.

During synchronization, the changes are also transferred to other attached documents (as required). For instance, during attachment of assembly drawing and explanatory note to the BOM, the data transfer was set up as follows: the BOM receives the designation and the name of product from the drawing, and the explanatory note receives them from the BOM. Later the designation of the product in the drawing was changed. During synchronization, the new designation will be transmitted from the drawing to the BOM, and from the BOM to the explanatory note.

The document, to which the changes are transmitted, cannot be loaded during synchronization (in this case it will be opened, modified and closed in background mode without being displayed).

After completing the synchronization, the system will generate a message enumerating the documents that have been modified.

6.1.3.6.4. Data Synchronization during BOM Opening

During the opening of the BOM document, a checkup is performed of the correlation

- between the BOM items in the BOM proper and the BOM items in the attached documents,
- and of the designations and names of the products in the BOM and in the documents attached to it.

If the system reveals some mismatch between the parameters of the same item in the BOM and in the model/drawing, or mismatch between the structure of the BOM items in the documents, a relevant message will be displayed (Figure 6.1.6).

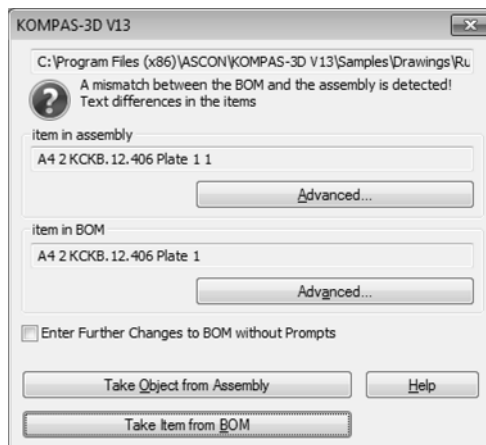


Fig. 6.1.6. Message on mismatch of BOM items

It contains information about the nature of the mismatch (e.g. “the text portion of the item differs in the BOM and in the drawing” or “the item is present in the drawing, but is absent in the

BOM”) and about the buttons to select the variant of mismatch elimination — **Take Item from Assembly** or **Take Item from BOM**.

General principles of automatic synchronization:

- Synchronization is performed even if the data transfer from a drawing to the BOM is disabled in the BOM settings.
- The user has the option of selecting a mismatch elimination method for each specific BOM item.
- The user can select a variant of mismatch elimination which is common to all mismatched items (transfer all items from a drawing to a BOM or from a BOM to a drawing).

A mismatch between the designation and the name of product in the BOM and those in the attached document may occur if a two-way link has been established between the title block of the BOM and the attached document (i.e. document transfers data to the BOM title block, and receives it from the BOM). For instance, a warning is displayed during BOM opening (Figure 6.1.7), if a document attached to the BOM via a two-way link is inaccessible or has a write-protection on at the time of the link saving.

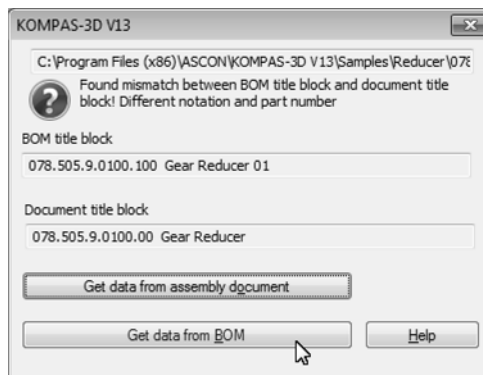


Fig. 6.1.7. Message on mismatch of product designation and name

The designation and the name contained in the BOM title block and in the document are shown in the relevant preview lines.

If the designation and name of the product in the BOM must remain the same as they are in the connected document, click the **Get Data from Assembly Document** button.

If you wish to leave the designation and the name in the BOM as they are, click the **Take Data from BOM** button.

6.1.4. Integration with Application Libraries

Some application libraries of the system KOMPAS-3D are fully integrated with the BOM designing system.

This is demonstrated by the fact that the insertion of a library image or a model into a drawing or an assembly causes the relevant BOM item to appear in this drawing or assembly.

An example of such library is the Library of Standard Items (hereinafter - the Library). When using it, the following issues shall be accounted.

- BOM item automatically appears in the document when an item is inserted from the Library, if special setting is made in it (default variant). You can enable/disable the option while inserting an object. Control of settings is described in detail in the Library help system.
- The library is set up by default, so that the designations of products and materials are formed in strict compliance with the relevant standards, and the BOM item generated during the operations with the Library are placed in the proper sections of the standard BOM.

You can change default settings — set up inclusion of an object in a specific BOM section or subsection, inserted or additional section block, or fillable BOM columns. For this purpose, a user template for BOM items should be created in the Library.



The system attribute library *Spc.lat* is used (see Section 6.3.5) to form templates for names of standard products and materials.

After insertion, you can edit text or modify values in the completion template fields. You can also set up sorting by a parameter (see Section 6.1.2.5).

- During the insertion of a library item into a drawing, you can create a new reference leader or specify an existing reference leader for displaying the inserted product (material). This leader, in the same way as an image inserted from the Library, will be automatically transferred to the geometry of the relevant BOM item (with all ensuing consequences — option for zone calculation, position number transfer from the BOM to the leader etc.)
- If a number of library items with similar parameters are inserted in the document, duplicate BOM items will be generated (“twin” items). For more details, see Section 6.2.6.15.
- Information contained in the Library can be transferred to the BOM document directly (not only from attached drawings). Obviously, the BOM items generated in this way cannot incorporate geometry. In the other respects they do not differ from analogous items which have been received from the drawings: their text portion is formed on the basis of a template, they are located in the same sections of the BOM and are sorted in the same way.

For details about the insertion of items from the Library to the BOM see Section 6.2.2.4.3.



Transferring information about BOM items to documents, as well as direct insertion of objects into BOM, are possible from any user application library, as long as it is properly set up.

In addition to the aforementioned Library of Standard items, a Materials and Assortments Library is integrated to the KOMPAS BOM. This integration manifests itself in the possibility for the insertion of material designations from the library into the BOM and viewing material properties (physical and mechanical, process-related properties, chemical composition, etc.) The procedure for the insertion of material designations from the library and its adjustment for work with the BOM are detailed in the User Manual for Materials and Assortments Library.

6.2. Methods of Working with the BOM

6.2.1. Interface

The BOM design system is running in the KOMPAS-3D environment. Thus elements of its interface (menus, panels, etc.) and methods of working with them are similar to those in KOMPAS-3D.

6.2.1.1. Property bar

If any object is selected in the table its parameters will be displayed in Properties bar (Fig.6.2.1).

Each parameter is corresponded with the field or the switch (see Section 6.2.2.6 on p. 1484). Having changed the switch mode and values in fields you may edit BOM object parameters.



Fig. 6.2.1. The BOM item Properties bar

6.2.1.2. Panel Current status

In the BOM operation mode the panel **current status** (Fig.6.2.2) displays fields and switches allowing to control parameters of the BOM current section (see Table). 6.2.1).

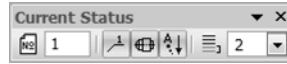







Fig. 6.2.2. **Current status** bar

Tab. 6.2.1. Controls located on the panel Current status when working with the BOM

Interface item	Description
 Current Page	This field displays the BOM current page number. To go to the required page enter its number in the field.
 Set positions	This toggle button allows permitting or prohibiting item numbering in the current section (see Section 6.2.2.8 on p. 1491).
 Attach Geometry	This toggle button allows permitting or prohibiting connection of the geometry to the BOM items in the current section (see Section 6.2.2.5 on p. 1482).
 Automatic sorting	This toggle button allows permitting or prohibiting automatic sorting of BOM items in the current section (see Section 6.2.2.7 on p. 1491).
 Quantity reserved lines	This field shows quantity of reserved lines in the current section (see Section 6.1.2.3 on p. 1452). You may change this quantity.

6.2.1.3. Keys of command call


 Buttons of command call of the BOM Design System are located on the **BOM** toolbar (Fig. 6.2.3). If it is included into compact panel use toggle button **BOM** to activate it.



Fig. 6.2.3. **BOM toolbar**

In the following sections of this manual buttons are shown on the margins of paragraphs with respective commands mentioned. Here toolbar name **BOM** is not defined.

The panel name is specially defined if the button is on the other panel.

6.2.2. Working with document-BOM

6.2.2.1. Creation of document-BOM

 To create BOM, call command **File — Create**. In the displayed dialog of new document creation select type of the new document— **BOM**.

A new BOM table will appear on the screen. You may create new BOM items in it (please, see Section 6.2.2.3).

This table and its filling in rules (columns, sections, availability of automatic sorting, quantity of reserved lines, etc.) will correspond to the style defined by default for a new BOM.

The BOM style may be changed.



You may create a new BOM based on the template. For this activate tab **Templates** of the new document creation dialog and specify required template.

Templates of BOM delivered with KOMPAS-3D contain the BOM style. You may edit existing templates (for example, by filling in master records in them) or to create new ones.

6.2.2.2. Selection of the BOM style

BOM style — parameters and settings of the BOM that affect its filling in and displaying. For more details about it, see Section 6.3.1 on p. 1543.



BOM Styles delivered in KOMPAS-3D distribution disk are stored in files *Graphic.lyt* and *Vector.lyt* from folder ...Sys .

6.2.2.2.1. Current BOM

Make the following to change the current BOM style.

1. Call command **Service — Parameters...**
The dialog will display by default section **Current BOM— Style**.
2. Click the **Libraries...** button and select *.*lyt* file where required style is located. You may not select the library over again if the dialog reference field already contains the required library.
3. Select the style for the current BOM from the list.



Usually the style of just created BOM with no object contained is being changed.

6.2.2.2.2. Newly created BOM

Make the following to set the default style for new created BOM.

1. Call command **Service — Parameters**.
2. In appeared dialog expand section **New documents — BOM — Style**.
3. Specify style library (file *.*lyt*) and select the style for newly created BOM.

6.2.2.3. BOM filling in

A BOM is filled in by creating new object in it (both main and auxiliary).

BOM items may be also created in connected 3D assemblies or assembly drawings and transferred then to the document-BOM.

Besides that, you insert objects from the Library of Standard Items (see Section 6.2.2.4.3 on p. 1477) or Materials and Assortments Library.

6.2.2.3.1. Adding new object



To create main item of the current BOM, call the **Insert— Main item** command or press *<Insert>*.



To create auxiliary object, call **Insert— Auxiliary Object** command.

If created object—is the first one in the BOM...

The dialog of section (and subsection) selection will display to locate this object (Fig. 6.2.4).

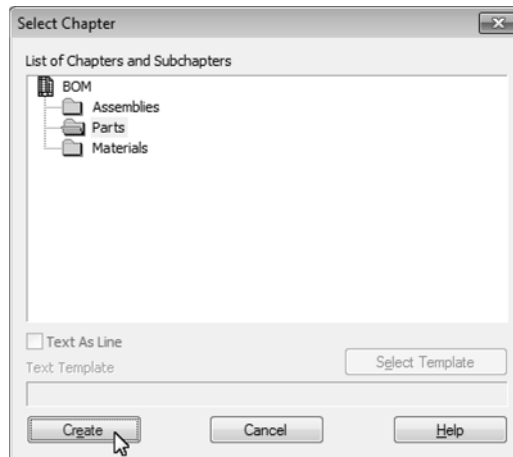


Fig. 6.2.4. Dialog for selecting a BOM section

Select required section in this dialog.

Title of the selected section will appear in the BOM blank form. The title will be underlined and framed with blank lines according to the standard.

If an object is required to be created in the current section...

To create an object in already existing section place the cursor in any place before calling command **Insert— Main item**.



If an object is required to be created in a new current section...

To create an object in a new section, first create this section using command **Insert— Section**. Since BOM can't contain empty section (section without objects), the first object will appear in the new section right after its creation.

Irrespective of the method of BOM item creation, after command calling the line to enter its text part will become accessible in BOM table (please, see Section 6.2.2.4).



When creating a new object in some sections (for example, *Standard parts*, *Materials*) it is required to specify method of its text part generation: using of text template (please, see Section6.2.2.4.5) or input of text part with keyboard (Please, see Section6.2.2.4.6).

After entering of text part (and if required,— also additional parameters, please, see Section6.2.2.6) confirm its creation. For this press key combination `<Ctrl>+<Enter>` or click the mouse outside the line containing the object.

6.2.2.3.2. Copying of BOM item

BOM item may also be created by copying of already existing object and further editing of made copy.

To copy an object, select it (not triggering to its text part editing mode) and call command **Editor— Copy object**.

The system will create a new object with the text part similar to that of the selected object and will switch to this text part edit mode (please, see Section6.2.2.4). After data changing in columns of the new object confirm its creation.

The object created by copying of the other object has the properties and parameters which do not differ from the objects created with the command **Insert— Object**.

Usually copying creates an objects with the text part which slightly differ from the text part of the already existing object.

Copied object will be located in the section where its prototype is located. Placing of the object copy in the other section is impossible.



When copying note that data from additional columns of the selected object (please, see section6.2.2.6.2) are transferred to the additional columns of the copy.

6.2.2.3.3. Creation of variants of BOM items

One more way of BOM item creation is to create a variant of the current object. Do the following to create a variant item.

1. Highlight main item of the BOM and call command **Insert— Variants**.
2. In displayed dialog enter numbers of object variants (Fig. 6.2.5).

The numbers must be separated by commas and dashes. The variants numbering may include skipped numbers.

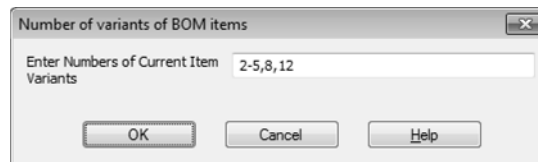


Fig. 6.2.5. Dialog of variant number entering

The result will be generation of new items in BOM which excel only in suffixes from their prototype in designation (we will call them **items-variants**).

For example, if we enter numbers of variants «2-5,8,12», 6 new items will appear in BOM, their designation will excel from item designation for which variants are created with suffixes -02, -03, -04, -05, -08 and -12.

The display of designations and positions of versions depends on the setup of the current BOM. To preview and/or modify this setup, call **Format – BOM settings** and activate the **Blocks of variants** tab in the BOM setup dialog that opens (Fig. 6.2.6).

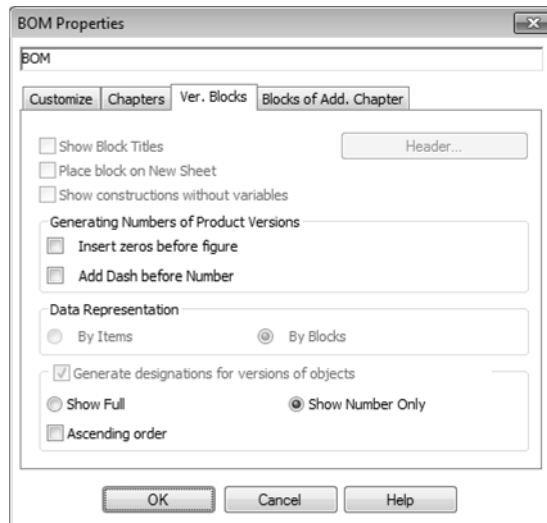


Fig. 6.2.6. Setting of object variant creation

The display of designations and positions of designations is controlled by the group of elements **Generate item designations**.

- ▼ If **Show full** is enabled, designation of new items consist of a designation of the initial item with an added suffix, i.e. version number (Fig. 6.2.7).

Format	Zone	Ref. No	Designation	Name	Qty.	Note
				<u>Parts</u>		
		18	AEKT.640820.129	Fixator	2	← Source object
			AEKT.640820.129-02	Fixator	2	← Object version
			AEKT.640820.129-03	Fixator	2	← Object version
			AEKT.640820.129-04	Fixator	2	← Object version
			AEKT.640820.129-05	Fixator	2	← Object version
			AEKT.640820.129-08	Fixator	2	← Object version
			AEKT.640820.129-12	Fixator	2	← Object version

Fig. 6.2.7. Items-variants. Designation is fully shown

- ▼ If **Show only number** is enabled designations of new items also consist of a designation of the initial item with the version number added, but constant part of designation in the BOM table is not shown — it shows only variant number (Fig. 6.2.8).

Format	Zone	Ref. No	Designation	Name	Qty.	Note
				<u>Parts</u>		
		18	AEKT.64.0820.129	Fixator	2	← Source object
			-02	Fixator	2	← Object version
			-03	Fixator	2	← Object version
			-04	Fixator	2	← Object version
			-05	Fixator	2	← Object version
			-08	Fixator	2	← Object version
			-12	Fixator	2	← Object version

Fig. 6.2.8. Items-variants. Designation shows only variant number

Items are automatically sorted only with full designation taken into consideration. Due to this items-variants are located in numerical order after its initial number even if only their numbers are shown in *Designation* column.



Objects-variants are created only in those sections of BOM which are sorted out by designation.

- ▼ If **Positions increase** is disabled, **Do not show position** is automatically enabled and **Item position increase** is disabled in settings of BOM items created by adding an item version. The **Item is not a version** option is always automatically enabled in settings of version items. See Section 6.2.2.6.1 on p. 1485 for BOM item settings.

6.2.2.3.4. Deletion of the BOM item

To delete a BOM object focus on that object (by clicking the mouse once or using the arrow keys). Then call command **Editor — Delete object**. Query to confirm item deletion will be displayed (Fig.6.2.9). Designation and name of the item will be defined in it.

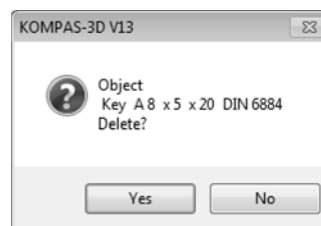


Fig. 6.2.9. Object deletion request

Click the **Yes** button, if you are sure regarding the object deletion, or the **No** button, if the object should not be deleted.



You cannot undo the deletion of a BOM object.

Be especially attentive to deletion of the BOM item during setting of the current BOM with enabled option **Delete geometry with deletion of the BOM item** (Please, see Section 6.2.6.1.1 on p. 1518), since it will result in deletion of all items included in the scope of the BOM item from assembly drawing and assembly model.

6.2.2.4. Input and editing of the text part of the BOM item

Text part of the BOM item may be filled using different ways.

6.2.2.4.1. Manual data input

Mode of entering and editing of the text part is enabled after you call command of a new BOM item insertion (please, see Section 6.2.2.3.1) until the item creation is confirmed.

The input and editing mode of the text portion has practically no differences from the text mode in KOMPAS-3D. In this mode a line will be opened in the BOM table for editing where the BOM item text will be located. Commands of font and paragraph editing, insertion of fraction, index, over- and underline, special symbol, symbol and predefined text will become available.

For symbols entered into a column, their horizontal spacing compression is automatically adjusted to fit them all in one cell. If contraction is too high you may transfer part of symbols to the following line. For this purpose, press `<Enter>` key in the place of transfer. Thus, text part of the BOM item may take several lines in the BOM blank form.

Moving of the text cursor inside the column is made using `<▶>` и `<◀>` keys. **The** `<S>` key moves text cursor to the position before the first symbol on the column, and `↑` key — to the position after the last symbol.

Moving between columns is made with `<Tab>` key (from left to right) and `<Shift>+<Tab>` key combination (from right to left). You can also enter the desired column of a BOM object by clicking it with the mouse.

To confirm the BOM text part changes and exit from its edit mode, press `Ctrl>+<Enter>` key combination in any part of the window outside of lines with edited text part.



Composite designations (i.e. designations which include, in addition to the basic part, the designation number and/or the additional number and/or the document code) are edited in a special dialog. This dialog automatically appears on the screen after an attempt to enter/edit a designation in the *Documentation* section (because in this section, designations usually include the document code) and after an attempt to edit a designation of a version item. Working in it and the result of the designation's entry are the same as when entering the title block; see Section 3.5.3.1.1 on p. 1156.

6.2.2.4.2. Input of codes and names of documents

When creating items in *Documentation* section it is often required to enter codes and names of documents, for example, “SB” code and “Assembly drawing” name. You may select codes and names from a special dialog in order not to do it manually.

For this call **Enter code and name...** command from shortcut menu in any column of the created item. The dialog will display to select document code and name. Working in this dialog and the result of code and name insertion are the same as when entering the title block; see Section 3.5.3.1.5.

Code and name are inserted in columns with *Line* data type, related with *Part name* and *Document designation* stamp cells. Data type in column and cell name are defined for connection for every section during setting of the BOM style. BOM styles delivered with KOMPAS-3D distribution disk are adjusted to allow insertion of code and name in all sections but *Standard parts*, *Other parts* and *Materials*.

6.2.2.4.3. Insertion of objects from the Library of Standard Items

In order to insert an object from the Library of Standard Items (hereafter, the Library), do the following.

1. Call the **Libraries — Standard items — Insert element command in the BOM**. The option of BOM item creation should be enabled in the Library's settings (by default).

Choose the required object in the Library (bolt, screw, nut, bearing, joint and so on). The selection of objects is described in detail in the Library help system.

2. Close the library window by clicking the **OK** button.

BOM item will be positioned in the BOM section defined by the library. For example, a fastener will be added to the *Standard items* section. If there is no such a section, it will be created automatically during the object insertion.

Text part of the item will be generated based on the filling template with the object parameters selected from the Library. Text part may be further edited (please, see Section 6.2.2.4.7 on p. 1480).

3. After insertion of the first library object, a Library window will appear on the screen for insertion of the next object. Insert the next object, if necessary.

You can insert several identical objects. In this case a duplicate object will be generated (see Section 6.2.6.15 on p. 1534).

In order to finish insertion, close the library window.



You may not use duplicate items if separate BOM objects are not required for every library object. In this case, insert only one item from the library and then enter required value in *Quantity* column.



You can insert object to a BOM from another application library, including a user one, as long as they are properly programmed. Insertion should be made in a way consistent with each library settings.

6.2.2.4.4. Editing of the BOM item text part

To enter the existing BOM item text part editing mode, double click the mouse on the lines with this item located or highlight the item and call **Editor— Edit item** command or press *<Enter>* key.

Exit from the text edit mode is made in a similar way as exit from the item input mode — by clicking the mouse in any place of the BOM outside of edited item or using *Ctrl*+<Enter> key combination.

6.2.2.4.5. Using filling templates

The text portion of some objects can be created by filling templates. The template is usually used to fill designations and names of standard items and materials. In general case templates are used to automate sorting of the texts generated based on standard rules.

Filling in template is a “blank” with margins to enter values of item properties. Text in a BOM object column is automatically generated from the data in the template fields. A template also stores information about which fields and in what order to use for sorting objects of a certain type.

BOM styles delivered in the system are adjusted to create items in their sections *Standard parts* and *Materials* based on the respective filling in templates. Let’s consider text part input based on the template for item— standard part.

1. When creating an item specify the section for its storage — *Standard items* (Fig. 6.2.10).

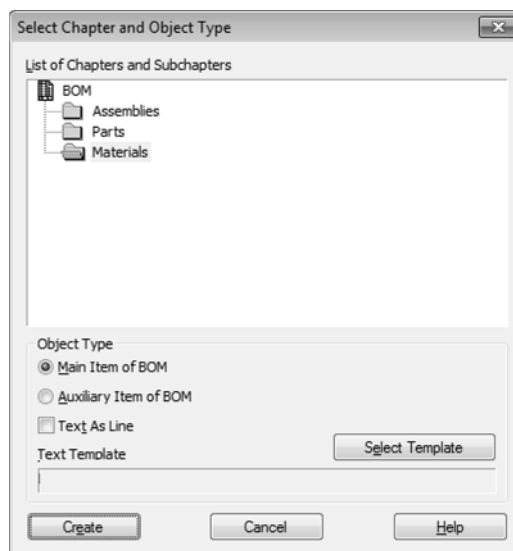


Fig. 6.2.10. Selection of Standard items section

2. In the same dialog click the **Select template** button.
Dialog of the template selection to fill in text part will be displayed (Fig.6.2.11).
3. Select the desired template in it.



Templates of the objects which may be created in this section are only available. For example, you can’t define template of any material in *Standard items* section, but you may select filling template of fastening part.

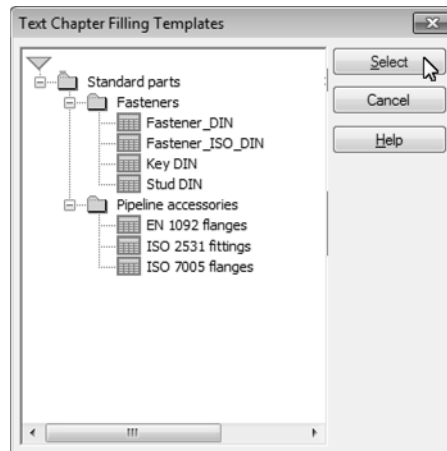


Fig. 6.2.11. Selection of filling template for text part

After confirmation of template selection a main BOM item will be generated. The item name generated based on the template (according to the standard for the selected part view) will be transferred to its *Name* column. Values of the item parameters (for example, diameter and length of the bolt) are used by default.

4. Manually enter required data in the remained item columns (for example, quantity of bolts) and, if required, edit the name generated based on the template (for example, change nominal dimensions of the part). Please see Section 6.2.2.4.7 for detailed description of the text part filled in according to the template.
5. To confirm item creation press `<Ctrl>+<Enter>` key combination.



If a BOM object is created by insertion into a graphic document of an object from the Library of Standard Items or Materials and Assortments Library, you do not need to choose a template. Text part of a BOM object is automatically completed in accordance with the template designed for the object (see Section 6.1.4 on p. 1467).

Text part of items may be filled on the template not only in *Standard parts* and *Materials* sections of standard BOM but also in other sections of BOM of any styles (including user one). You may use not only templates of filling in of names of parts and materials delivered in KOMPAS-3D but also those templates of any type created by users (for example, template of part fonts or template of enterprise codes). Please, see how to create user BOM templates and styles in part 6.3.

6.2.2.4.6. Template text part as a line

When creating an item in *Standard items* and *Materials* define a template of the text part filling in (please, see Section 6.2.2.4.5).

Sometimes it is required to enter an object with no ready template for it, the item text part of which is generated using the template.

If you have to enter such objects quite often it is obediently to create a special template for them and to store it in the library *Spc.lat* (see Section 6.3.5 on p. 1563).

But if such items are rare, one can do well without a special template. In this case when creating an item in the dialog of selection of section and item type (Fig. 6.2.10), enable option **Text part as a line**.

Enter text part of the BOM item and confirm its creation. Entered text part will automatically be split in fields and sorted out by these fields. Sorting order may be changed if you manually edit keys of these fields (please, see Section 6.3.5.1 on p. 1563).

6.2.2.4.7. Editing of text part filled in based on the template

If text of any column of the BOM item was generated based on the filling template double clicking the mouse in this column in edit mode will display a dialog where you may change values and texts in template fields (Fig. 6.2.12).

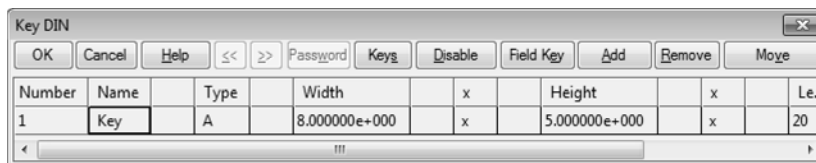


Fig. 6.2.12. Dialog of value editing in template fields

After confirming the changes to the data in the template, the text in the column is re-created per the new values in the fields.

The text portion of a BOM object can always be edited without changing the values in the fields of its template. To do this, you need to enter and delete data in the mode of editing the text portion, without calling the dialog of editing values in the template fields.



In any case, when sorting objects that use a filling template, the respect is given to the data in the template fields, rather than to the visible text portion in the BOM table.

6.2.2.4.8. Using of predefined texts

When entering the text portion of BOM object, it is possible to use text templates. You may work with them similarly to working with text templates in text and graphic documents.



You should distinct text templates and filling templates.

Predefined texts to insert in BOM may be stored in a separate file of templates **.tdp* (not necessarily in the same file of text templates which is used by graphical and text documents). The name of the text templates files from which templates are inserted, may be specified in the BOM adjustment dialog (see Section 6.3.2.1 on p. 1547). By default predefined text file specified in the current BOM style is used.

To insert predefined text into the BOM item column in the text part edit mode, click the mouse in this column or placing the cursor over it call command **Insert text** from the shortcut menu.

In displayed Text template librarian select required text and double click it with the mouse.

If during setting of the system the section **Text editor – Text templates** has **Save entered text in the file** option enabled, then texts entered manually into the item columns will be automatically stored in the text template file. The structure is generated in **BOM** section of this file which considers sections and columns of the BOM (Fig. 6.2.13).

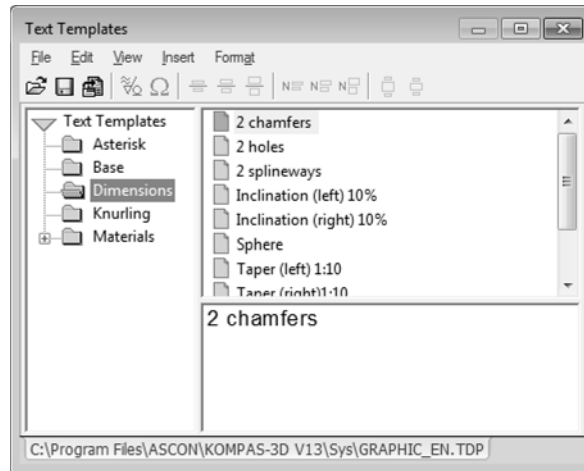


Fig. 6.2.13. Text templates

6.2.2.4.9. Semiautomatic data input into the BOM graph

If KOMPAS-3D document is connected to the BOM item (see Section 6.2.2.6.3 on p. 1489), data from this document will be transferred to the BOM item text part (provided that such data transfer was enabled during the BOM style setting). In this case, there is no need in the manual data input into some of the object columns, which helps avoid typos. For example, data from the drawing title block connected to the object may be automatically transferred to the BOM columns *Format*, *Designation*, *Name* and additional column *Mass*.

After editing and saving this data in the connected document, they will be transferred again to the BOM object. Therefore, due to a relation between a BOM object and the document, the text portion of the object is filled in, and a concordance is maintained between the information in the title block of the connected document (for example, a part drawing) and the text portion of the BOM object.

If BOM item contains geometry from assembly drawing (see Section 6.2.2.5), including the leader, designation of the zone where the leader begins (provided that zone calculation would be included during adjustment of BOM style) will be automatically transferred to *Zone* column.

If during assembly drawing editing position of leaders included into the BOM geometry contents has changed then at the moment of position calculation, *Zone* column would receive new designations of zones where geometry of BOM items would be now located.



In any case you will have to manually enter data in some columns (for example, *Quantity* and *Note*) since the system will not be able to automatically generate them.



It is recommended to use semi-automatic data input in the BOM item text part as often as possible. This will allow preventing error occurrence during BOM generation.

6.2.2.5. Connection and editing the geometry contents of the BOM item

Insertion of position leader to the BOM item is a necessary condition for transferring information about BOM items between the drawing (or assembly) and the BOM.

Insertion of the respective objects to the BOM item allows you to quickly locate the image of this BOM item (for example, part or assembly unit) in the assembly drawing or model.

Therefore, it is certainly recommended to include the position leader into a BOM item along with its image or model, if possible.



The leader may be the only graphical object in the BOM item geometry contents.



You may include several position leaders in the BOM main item geometry. In this case the BOM item position number will be transferred to all connected leader shelves (they will have similar position numbers).



The same leader may be included into the contents of several BOM items. In this case it will be automatically added with additional shelves to place the number of every object.

6.2.2.5.1. Insertion of the geometry in the contents of the BOM item

Geometry from the drawing

The most quick and convenient way of geometry insertion into the BOM item contents is to create a BOM item in a drawing containing the geometry and to transfer this item to the BOM connected to the drawing. Please, see description of creation of the BOM item containing the geometry in Section 6.2.3.1 on p. 1494. BOM items are transferred from the drawing to the BOM related to it automatically during connection of one document to another one (please, see Section 6.2.4.1 on p. 1507), as well as during storage of the drawing connected to the BOM or manually using synchronization command (please, see Section 6.2.4.2.1 on p. 1513).

If BOM item is created without the geometry the latter may be inserted into the item contents as follows.

1. Select the drawing's graphic objects, which shall be added to the BOM object (do not forget about the Reference Number Leader!).
2. In the BOM connected to the drawing highlight the item into which contents the geometry should be entered.



3. Call the **Editor — Edit contents** command.

4. In displayed query confirm addition of graphical objects in the contents of the BOM item (Fig. 6.2.14).

After that, the selected geometry enters the object.

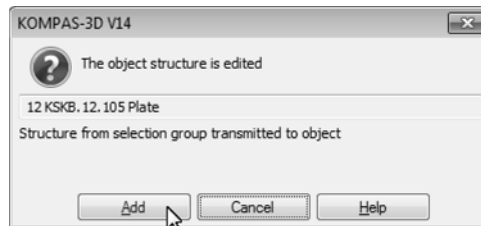


Fig. 6.2.14. Confirmation of changing of the BOM item geometry contents

Geometry From Model

One BOM item can include the geometry of either one component or one or multiple bodies.

Component geometry is included to the BOM item when creating this item (see Sections 6.2.3.3 on p. 1497 and 6.2.3.4 on p. 1497).

The bodies created in an assembly can be included into the BOM item geometry both at the moment of their creation (see Section 6.2.3.4.2 on p. 1498, description of method No.4) and afterwards. Adding a body to the BOM item geometry is identical to adding a graphical object to the BOM item geometry (see above). However, unlike graphical objects, one and the same body can be a part of only one BOM item. In other words, the BOM item can only include the body, which is not a constituent part of another item.

The position leaders existing in a model are included into BOM items in the same way as the position leaders of drawings.

6.2.2.5.2. Changing BOM Item Geometry Contents

... In assembly drawing

A situation frequently occurs when it is required to change a set of graphical objects contained in the BOM item contents— for example, to add graphical objects to the BOM item geometry or to change the BOM item geometry with the other geometry or to delete the geometry from the BOM item contents.

Sequence of these steps fulfillment is approximately the same.

1. Select graphical objects which should be included in the BOM item contents. Neither of graphical objects should be highlighted if the geometry is required to be deleted from the BOM item contents.
2. In the edit mode of the BOM connected to the drawing, highlight the item the geometry contents of which is required to change.
3. Call the **Editor – Edit** object contents command.
4. In the query appeared on the screen (Fig. 6.2.14) select a variant of changing of the BOM item graphical contents (**add** or **replace** geometry) or confirm deletion of the geometry from the BOM item contents.





Be attentive during changing of geometrical contents of the BOM item, since it will be impossible to cancel this operation.

... in model-assembly

If a BOM item includes an assembly component, the item geometry includes all the bodies belonging to the component. It is impossible to manually edit geometry contents of such BOM item.

If the BOM item includes one or several assembly bodies, the BOM item geometry can be deleted, supplemented or replaced identically to the operations performed in an assembly drawing (see above). Bodies may be added to the BOM item only if they are not parts of other BOM items.

The position leaders existing in a model are added to and excluded from the BOM items identically to the position leaders of a drawing.

6.2.2.5.3. Reviewing BOM Item Geometry

Do the following to review the BOM item geometry.

1. Open the BOM and the assembly drawing or assembly connected to it.
2. Tile those document windows.



Call command **Show all** in the drawing or model window.



3. Activate BOM window and highlight any BOM item in it.
4. Call the **Service— Show Set of Attached Geometry** command.

The system will switch to the BOM item geometry displaying. Graphical objects and 3D components included into the contents of the selected BOM item will be highlighted in the drawing or assembly window.

5. To review geometry of other items of the current BOM, highlight these items with any method (with the mouse or keyboard).

The BOM item geometry showing mode is enabled until **Show item contents** button is pressed and the name of **Show item contents** command has the check box selected.

6. To disable geometry showing mode, release **Show item contents** button or recall **Show item contents** command.

6.2.2.6. Input or editing of additional BOM item parameters

To work with all types of additional parameters of the current BOM item, use controls from Properties panel **Parameters** and **Documents**.

After highlighting of any BOM item in these tabs the switches and fields are activated that allow defining additional parameters of this item. To change any parameters activate or disable respective switch or enter required data in the field.



Some fields of Property panel are for reference. It is impossible to change data in them.





6.2.2.6.1. BOM item settings

Item settings affect its position and reflection in the BOM and define item processing rules during fulfillment of some commands. Elements of Table 6.2.2 are used to control settings.

Tab. 6.2.2. BOM item setting controls

Element	Description
Type	<p>In this reference field type of the line highlighted in the BOM table is displayed:</p> <ul style="list-style-type: none"> ▼ Main item (see Section 6.1.1.1.1 on p. 1447), ▼ Auxiliary item (see Section 6.1.1.1.2 on p. 1448), ▼ Section header (see Section 6.1.2.1.2 on p. 1452), ▼ Empty line (see Section 6.1.2.2 on p. 1452), ▼ Reserved text line see Section 6.1.2.3 on p. 1452), ▼ Variant block header (see Section 6.1.2.4 on p. 1453).
Section	This reference field displays the name of the current BOM section.
Subsection Name	<p>These fields are used to move the item in the other subsection inside the section to which it belongs. Values in them are interrelated: when selecting subsection name in the list, the number of this sub-section will appear in the neighboring field and vice versa, if you enter the number of this subsection you will see the name of the respective subsection.</p> <p>Position of the object may be changed if you select the other subsection.</p> <p>Lists of names and numbers of subsections will be empty if the current section has no subsections.</p>
Number of subsection	
Ref.No. of object increases*	
Show item in table*	This switch affects displaying of the item in the BOM table (makes it visible or invisible). Disabling of the item will not delete this item but makes it invisible (up to enabling) in the BOM table **.
Show position*	This switch enables and disables displaying of the BOM item position number in the table.

Tab. 6.2.2. BOM item setting controls

Element	Description
	<p>Item is Variant*</p> <p>This switch automatically activates in objects generated with command Add item variants (please, see Section 6.2.2.3.3). This switch has principal meaning only when making group BOM with the number of variants of more than ten. Enabled switch means that when displaying information in the BOM by items the item should not be separated from its initial item. The first previous item with similar designation with disabled flag An item is a variant is considered to be an initial one.</p> <p>Besides for items with this flag enabled, Add variants command is unavailable (i.e. it is impossible to create a variant of the item-variant).</p>
	<p>Synchronize with document-container.</p> <p>This switch automatically activates in object with which an assembly component is linked. This component is a document-container of the BOM object. While the switch is enabled, transfer of changes to the BOM item is performed between an assembly (or BOM to which it is connected) and a document-container. If the switch is disabled, there is no transfer of changes. For example, you may need to switch off data transfer to modify a BOM item whose document-container is unavailable for recording.</p>
	<p>Reset</p> <p>This button is accessible if the user has changed switch statuses automatically set during creation of the BOM object. It allows resetting automatically defined settings.</p>
	<p>Locate on the new sheet</p> <p>The switch allows placing highlighted item or section header on a new BOM page.</p> <p>The switch is not accessible if the BOM is placed on the drawing.</p>
	<p>To bring text part to the initial view</p> <p>This switch is accessible if the BOM item appeared during insertion of fragment or library model into document (please, see Sections 6.2.3.1.2 on p. 1495 and 6.2.3.4.4 on p. 1500), and text part of the item was edited manually. Switch allows returning text part of the item “arrived” together with the insertion, i.e. to cancel manual editing of the text part.</p>

* Combination of switches **Item position increases**, **Show object in table**, **Show position**, and **An item is a variant** controls the item “behavior” during automatic position marking and item sorting.

** When working with the BOM you may enable the mode in which all BOM items are visible (and may be printed out) (both visible and hidden using **Show object in table** switch. For this call command **Show all objects** (please, see Section 6.2.6.7 on p. 1522).

Status of push-buttons **Position increases**, **Show item in the table**, **Show position**, and **Object is a variant** are set for every item at the moment of its creation. It depends on the way of the item creation and its characteristics.

Due to this during sorting, defining of positions, splitting of the BOM with big number of variants into blocks all items will behave according to their characteristics.

Let's consider several examples of automatically generated combination of settings of main item. In these examples it is intended that positioning in the BOM sections is allowed (enabled).

- ▼ Object is created using **Insert — Main item** (see Section 6.2.2.3.1). Its text part in the column which is sorted out differs from the texts in this column of the other items of the section.

For example, an object is created in *Parts* section the designation of which doesn't match to the others (already existing in the BOM) parts.

At the moment of confirmation of the item creation the system will automatically make sorting in the current section: compares all text in column *Designations*, which was sorted out. "Making sure" that designation of the new object is unique (i.e. there is no such part in the BOM yet), the system will activate switch **Item position** for it. It means that the item will have its own position number that differs from the others. Also switches **Show object in table** and **Show position** are activated (the system has no reason to automatically hide a unique BOM item of the BOM or number of its position).

Thus, a new item will be shown in the BOM table and every time during positioning it will be assigned with the position number different from the number of the position of the previous numbered item.

- ▼ The item is created by copying of the other item (please, see Section 6.2.2.3.2). Its text part in the column which is sorted out doesn't differ from the texts in this column of the initial item. Or the item is created with **Insert— Main item** command and its text part in the column which is sorted out doesn't differ from the text in such column of the other item of the section.

For example, an item with the text part which repeats the text part of already existing object-detail created in *Parts* section¹. At the moment of the item creation confirmation the system will automatically sort the current section. Having discovered that the part with such designation already exists in the BOM, the system places a new object-part under the existing object with similar designation. The system "understands" that the number of the new item position should be similar to that of the previous item (because they describe the same part) and that is why it disables **Item position increases** flag. The same part should not be mentioned in BOM twice, that is why **Show item in table** flag will be automatically disabled. Show position **flag** is not switched off. Due to this in the mode of showing of all items (please, see Section 6.2.6.15 on p. 1534) the number of the hidden item position will be visible.

Thus new item will not be shown in the BOM table, and every time during position marking it will be assigned with the position number similar to the previous one.

- ▼ An item was generated with **Insert — Variant** command (please, see Section 6.2.2.3.3). Irrespective of the fact whether only displaying of the variant or full designation is enabled, full designation is stored inside the object. Sorting is made right after the item creation and according to this designation the item will be placed right after the initial item with the same designation but without the suffix. The item has switches **An item is a variant** (due to this the item "remembers" its full designation) and **Show item in table** (based on a new designation the BOM "understands" that there is no such item in the section yet).

1. This may be required to connect geometry from different page of assembly drawing to the same part in the BOM.

Generally speaking the necessity to assign a new position number to the item-variant and displaying of this position in the table depend on the special case of the item-variant using. For example, if various variants of one part are used in one assembly unit, *Position* column of the items corresponding to them should show different numbers.

If different variants of one part are used in different variants of the assembly unit, they should have the same position number in group BOM which is shown only for the main variant.

BOM may realize both numbering methods of items-variants.

- ▼ If during BOM setting (**Format— BOM setting— Variant blocks**) **Positions increase** option is disabled (please, see Fig.6.2.6 on p. 1474), during creation of items-variants switches **Item position increases** and **Show position** are disabled. The result is that items-variants are found not numbered and the item that follows after them has position number that is more by 1 than that of initial item for which variants were generated.
- ▼ If during setting **Positions increase** option is enabled, then during creation of items-variants they have **Item position increase** and **Show position** switches activated. The result is that items-variants are found numbered in ascending order.



If item-variant is the first item of the section, i.e. there is no main item above it or the other variant with the same designation, this item position will be shown in the table independently on **Show position** switch status.



Mark positions button action on panel **Current status** covers only newly created BOM items. If item-variant was created with pressed **Mark positions** button, disabling this button will not result in hiding or changing of its position number. You may hide position number (or make it non-ascending) only using Properties panel switches. But all items created after disabling of **Mark positions** option will not be numbered.

6.2.2.6.2. Additional columns

A BOM item may contain data supplementing the information included in the standard form. This information is stored in so called “additional columns” of BOM item. They may be reviewed or edited at any moment, but they are not visible in blank form and are not printed.

An example of information in the additional columns is an item’s mass or price.

Any other item-related information may be also included in additional columns (OKP Code, material, text comments, etc.). Their quantity and contents are defined by user demands and stored in BOM style (please see Section 6.3.1 on p. 1543).

Data in additional columns of BOM item may be seen and edited in drop-down panel **Additional columns** from **Parameters** tab (Fig. 6.2.15).

Add. Columns	
Name	Value
Mass	4,275
Price	1250

Fig. 6.2.15. Additional Columns

The panel contains the list of all possible for this BOM types of additional columns and their values pertaining to the current BOM object.

To enter and edit values in additional column click the mouse in the respective field of the list and enter required data.

The utility commands allow summing up the numerical values of the auxiliary parameters; in this way, the number of identical objects in an assembly and even the number of objects in its different variants can be accounted for. Please, learn the order of calculation of value sum in additional columns in Section 6.2.6.2 on p. 1519.

6.2.2.6.3. Connection of document to the BOM item

BOM item may be connected with KOMPAS-3D document. This relation is two-sided and associative, and it allows transferring data about a BOM object into the connected document, or vice versa, the data from a document into its respective BOM object.

The document is connected to the BOM item in **Documents** tab of Properties panel. It contains expanding panel **Documents** (Fig. 6.2.16).

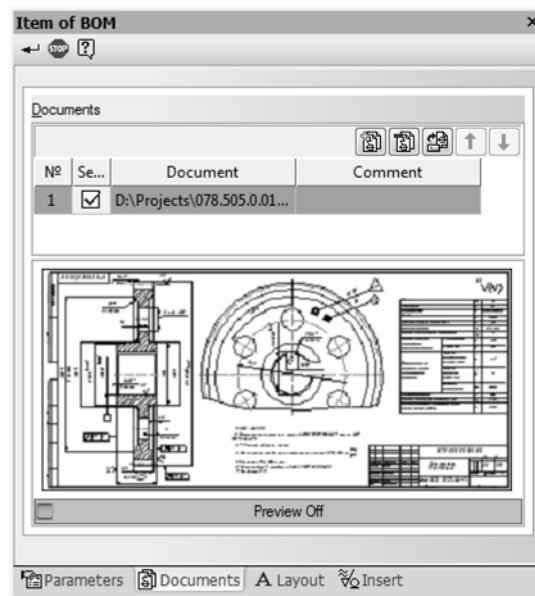


Fig. 6.2.16. Document connected to the BOM item

Panel contains name of the document connected to the BOM item and its icon image (provided that **Disable review** option is not enabled).



To connect the document to the BOM item on **Documents** panel click the **Add document** button. In the dialog that appears, enter the path to the document file and its name and click **Open**.



If the BOM item is a document, connect this document to it (for example, assembly drawing, explanatory note, diagram, etc.).

If the BOM item is an assembly unit, connect the respective BOM to it.

If the BOM item is a part, connect working drawing of this part to it.



A 3D model of the part or assembly is usually connected to the item automatically (please see Section 6.2.3.4.2 on p. 1498). It may be also connected manually (similar to documents of other types).

When a document is connected to a BOM item, a request for use of data from the title block of the connected document appears on the screen.

A positive response results in automatic transfer in the BOM item of the name and designation from the model or main text of the drawing as well as format designation in which connected drawing is made.

The information from a connected document can be passed not only into the columns of the BOM form, but also into the additional columns. For example, mass of the part from the model or from the respective column of main text may be automatically transferred to the additional column. At that, if the mass units used in BOM (see Section 9.2.5.1.4 on p. 1757) differ from the units used in a drawing or model, mass values will be recalculated.

Generally speaking, data transfer from the document connected to the item is the part of the BOM style. During style setting you may disable data transfer or set its parameters (for example, to change stamp cell, data from which should be transferred to the certain column of the BOM item).

Changes in the designation and name can be transferred in the reverse direction — from a BOM item to the connected document.

Data transfer (in both directions: from the BOM item to the connected document and back) can be enabled/disabled with **Transfer changes to document** next to the document's name in the **Documents** bar.

If this option is enabled:

- ▼ changes in the document connected to the item (designation, name, mass, sheet format) are transferred to BOM when the document is saved,
- ▼ changes in the designation and name in a BOM item are transferred to the document connected to the item when the item is saved.



To disconnect the document from the BOM item highlight its name in the list and click the **Disconnect document** button.



If the document connected to the item is required to be opened highlight its name in the list and click the **Edit document** button.



If required, several documents may be connected to BOM. In this case data of the first connected document will be transferred to the item. You may change the order of the documents connected to the item by moving them in the list using **Move document up in the list** and **Move document down in the list** buttons.



6.2.2.7. Item sorting

Automatic sorting of main items is made only in the BOM sections with enabled sorting (please, see Section 6.3.2.2 on p. 1550) and its rules are defined — sorting column and type.



To quickly enable/disable automatic sorting in the current section click the **Automatic sorting** button on the **Current status** panel. Pressed button means that automatic sorting is enabled in the current section, released button means that it is disabled.

Disabling of automatic sorting covers items newly created in the section and doesn't affect the order of already sorted items.

If automatic sorting is disabled in the section you may place its items in random sequence moving them up and down.

To move item in the section highlight it and call command **Editor — Move item up** or **Move item down**. You can use key combinations $\langle \text{Shift} \rangle + \langle \uparrow \rangle$ и $\langle \text{Shift} \rangle + \langle \downarrow \rangle$, respectively, for quick calling of the commands.

Automatic sorting of auxiliary BOM items is not made. They may be moved up and down manually even in the sections with enabled automatic sorting of main items.

If required you may sort out auxiliary items using subsections.

For this, move auxiliary items which should be sorted out to the beginning of the section— in order not to have main item before them. Then edit every of these items having assigned it required subsection number.

Items will be sorted out in ascending order of subsection number. Both auxiliary and main items will be considered.



If an auxiliary item adjusted with the above mentioned method is moved with **Move item up** and **Move item up** command to the place below the main item, a connection will occur between this main item and moved auxiliary item. Auxiliary item will be assigned with the same subsection number as the main one and will be moved together with it.

To disable this connection, move auxiliary item at the beginning of section and define required number of subsection for it.

6.2.2.8. Position marking

Numbers automatically appearing in column *Position* of new items are not item numbers. These are sequential numbers of items and reflect sequence of their generation.



To assign numbers of positions corresponding to their position in the BOM to items, call command **Service — Set positions**.

In the result of the command fulfillment numbers of item *positions* will appear in column.



To quickly enable or disable item number setting in the current section click the **Set positions** button on **Current status** panel.

Pressed button means that position setting in the current section is enabled, released button means it is disabled.

During positioning of reference numbers setting in each section, the number of reserved lines in the previous section is accounted for.

For example, *Parts* section ends up with 46 position and contains three reserved lines. Thus the section that follows it is *Standard parts* and it will start from 50 position, since position numbers 47, 48 and 49 are reserved for further changes.

Even after input of new or deletion of existing items in *Parts* section the number of reserved lines remains not changed in it and command of automatic position setting will provide “distance observance” between numbers of positions in neighboring sections.



For quick changing of the number of reserved lines in the current section, use **Quantity of reserved lines** field on **Current status** panel.

6.2.2.9. Modes of operation with BOM

When creating and filling in the text portion of objects in a BOM document, what you see on the screen is a standard BOM table, and you enter data in its entries. This working mode is called the **normal mode** of filling a BOM. It is similar to the normal mode of filling a text document.

In this mode additional parameters of BOM items are edited, geometry is connected to items, sorting is made, positions are set and other operations with items are made.

The title block of a BOM document is not shown in the normal mode and is not available for editing.

In order to complete the BOM's title block, view additional sheets, and enter text in them, you will have to switch to the **page layout mode**. When working with BOM this mode is similar to the text document page layout mode.



To switch to page layout mode call **View— Page layout** command or click the **Page Layout** button on the **Modes** panel.

In this mode, the BOM pages are shown just like they will be printed out. The title block tables of a BOM document and additional sheets' text are displayed and accessible for editing.

BOM objects are not accessible for editing in the print layout mode.

To edit items it is required to return to normal mode.



To switch from page layout mode to normal mode call **View — Normal mode** command or click the **Normal mode** button in **View** panel.

By default the system is in normal operation mode with the BOM.

6.2.2.10. Displaying of items containing code and name of document

Showing of item name in the text part of the BOM item containing code and name of the document, may be automatically disabled.

Disabling occurs if contents of *Designation* (without standard code considered) and *Name* (without standard document name considered) cells of the BOM item fully match contents of the respective cells of this BOM main text. Full text in *Name* cell of the BOM item is visible only in edit mode of this item text part.



List of standard codes and names of documents is defined by file of codes and names — *graphic.kds*. By default it is stored in \Sys subfolder of the main system folder. File format— is a text one that is why if required *graphic.kds* file may be opened and edited in any text editor. If non-standard code and/or name is used, contents of *Name* cell is displayed in full even provided that designation and name of the part match.

Let's consider mechanism of disabling of the part name displaying on the example of item of section *Documentation*. Let's suppose that the item corresponds to the assembly drawing of the part. It is obedient to connect assembly drawing to it and take data from its main text (Fig. 6.2.17).

					KSKV.12.020SB			
					Cantilever Assembly drawing	Lit.	Mass	Scale
Bord.	Sheet	№	Sign.	Date			6,146	1:2
Resp. Eng.	Ivanov			24.01.00				
Check. Per.	Ivanchenko			26.01.00		Sheet	Sheets	1
Mfg. Eng.						Konbers LTD		
Qual. Eng.								
Appr. Per.	Polikarpov			01.02.00				

Fig. 6.2.17. Main text of the assembly drawing connected to the BOM item

Lines “*KSKV.12.020SB*” and “*Supporting arm Assembly drawing*” will be respectively transferred to cells *Designation* and *Name* of the BOM item (Fig.6.2.18, a).

Let's columns *Designation* and *Name* of the main text contain lines “*KSKV.12.020*” and “*Supporting arm*” respectively.

In this case *Name* cell contents of this BOM item depends on operation mode. Part name — “*Supporting arm*”— is displayed only in this item edit mode (Fig. 6.2.18, a). After exit from this mode and also in page layout mode the part name image is disabled and only document name remains in *Name* cell (Fig. 6.2.18, b).

					<u>Documentation</u>		
*			KSKV.12.020SB		Cantilever Assembly drawing		*/A4

a) Edit mode

First-Time Usage					<u>Documentation</u>		
	*			KSKV.12.020SB	Assembly drawing		*/A4

b) Page layout mode

Fig. 6.2.18.

Let's now *Designation* column of the BOM main text contains «KSKV.12.021» line (Fig. 6.2.19, a). In this case *Name* cell of this item of the BOM displays full document name in any operation mode (Fig. 6.2.19, b).

					KSKV.12.020SB		
<i>Bord. Sheet</i>	<i>№</i>	<i>Sign.</i>	<i>Date</i>	Cantilever Assembly drawing	<i>Lit.</i>	<i>Mass</i>	<i>Scale</i>
<i>Resp. Eng.</i>	Ivanov		24.01.00			6,146	1:2
<i>Check. Per.</i>	Ivanchenko		26.01.00				
<i>Mfg. Eng.</i>					<i>Sheet</i>	<i>Sheets</i>	1

a) Main text of BOM

<i>First-Time Usage</i>					<u>Documentation</u>		
	*	KSKV.12.020SB			Cantilever		*A4
					Assembly drawing		

b) BOM item. Page layout mode

Fig. 6.2.19.

6.2.3. Working with BOM items in drawings and models

BOM items may be created not only in documents-BOM, but also in drawings and in models.

Methods of creation and editing of BOM items in different types of documents are practically similar. For example, copying and editing of main and auxiliary BOM items, creation of item variants, editing of additional parameters of items and connection of documents to them are made similar to the same operations with items in a BOM document (please see Section 6.2.2 on p. 1470).

Buttons of command call to work with BOM items in graphical modes of work with documents and models are stored by default on **BOM** toolbar (Fig. 6.2.3 on p. 1470).

6.2.3.1. Creation of BOM items in the drawing

Only the BOM items may be created in assembly drawings the images of which are contained in this drawing. As a rule these are items from section *Assembly units, Parts, Standard items*.

Do the following to create BOM item.

1. Being in the edit drawing mode highlight graphic items composing image of the part or subassembly including position leader. Selected items will compose geometry of the BOM item.



Insertion of position leader in the item geometry will provide automatic filling in of the column of *Zone* BOM and transfer of position number from the BOM to the leader shelf.



2. Call the command **BOM – Add item...**

3. In the displayed dialog select section to place an item and if required— template of its text part filling in.
4. Enter text part of the BOM item in the displayed window (please, see Section 6.2.2.4 on p. 1476).
5. If required, define additional item parameters in **Parameters** and **Documents** tabs of Properties panel (please, see Sections 6.2.2.6.1 on p. 1485 and 6.2.2.6.2 on p. 1488) and connect documents to it (please, see Section 6.2.2.6.3 on p. 1489).
6. Confirm creation of the BOM item by clicking the **OK** button in the text part input or key combination `<Ctrl>+<Enter>`.

An item containing selected geometry shall be created in the drawing. If **Position Numbers with BOM Items** option is enabled in the text elements color setup dialog, the position number font color shall change according to the settings. This allows you to distinguish between the positions included into BOM items and those not included. To call the dialog for text element setup, use the **Service — Options... — System — Screen — Color of text elements**.



If graphical objects (including position leader) were not included during item creation you may include them into the BOM contents at any stage of working with it. It is made in a similar way as when inserting geometry into the item of the document-BOM (see Section 6.2.2.5.1 on p. 1482). The only difference is that it is required to select an item with the geometry being entered in its contents, only in slave mode.

6.2.3.1.1. BOOM objects from the Library of Standard Items

When inserting an image from the Library of Standard Items (hereafter, the Library) into a drawing, a BOM object with a positional leader is created by default.

After selection of an object from the Library and specifying its position on the screen, a window for entry of the BOM item's text part will appear (similar to the one displayed during creation of the BOM item in the drawing).

After confirmation of object creation, a new positional leader should be created (default variant) denoting the object. This leader and graphical items of the library image will be included into the BOM item contents.

The BOM item will be placed in a certain section of the Library, for example, a fastener will be located in the *Standard items* section.

You may decide not to create a BOM item during image insertion from the Library or create the BOM item after its insertion into a drawing or fragment. Insertion of objects, BOM object creation, position setup are described in the Library's help system.



You can insert BOM objects from another application library in a way provided for by the library.

6.2.3.1.2. BOM items from inserted fragments

It is possible to create the same BOM items in the fragment as in the drawing. Those are created and edited by the same rules as BOM objects in a drawing.

Fragment is not connected to the BOM. Therefore, object cannot be directed passed from it into a BOM.

However, a fragment can be inserted in a drawing. The BOM items that are similar to graphical ones will come from the fragment to the drawing (with all properties stored). Such item transfer occurs in any insertion method — by link, body or exploded.

Order of operation with items made by such method doesn't differ from the order of operation with BOM items created directly in the drawing.

If position reference number leaders are part of the contents of BOM objects in a fragment, then, when inserting a fragment into the drawing, after sorting all BOM objects in this drawing the leaders will bear the new position reference numbers, according to the position of the objects that came from the fragment, among the drawing's BOM objects.

That is, if the fragment has position numbers *1, 2* and *3*, then after insertion of this fragment into the drawing and recalculation of positions in it the numbers may be, for example, as follows *10, 28* and *46* (provided that position leaders are included into BOM item contents, calculation of positions and their positioning are enabled).

If the fragment contains several descriptions of BOM (please, see Section 6.2.6.14 on p. 1534) and items created for every description, then when inserting this fragment into the drawing all descriptions containing in the fragment will appear as well as all BOM items from the fragment.



If during issuing of various assembly drawings (schemes, general view drawings, etc.) you use standard solutions, create respective fragments with BOM items and insert them in drawings using any method.

1. Draw standard part (or unit) in the fragment.
2. Create a BOM item in this fragment corresponding to the part (or BOM items corresponding to unit parts).
3. Save the fragment.
4. Insert the fragment into assembly drawings.

BOM items will “be coming” together with the image inside them.



You may also create libraries of fragments containing BOM items and insert fragments from them into drawings. When working in the library the fragments containing BOM items doesn't differ from other fragments.

6.2.3.2. Creation of BOM items in the model. General Information

There are two different methods of working with BOM items in models.

1. Manual creation of BOM items in component models (see Sections 6.2.3.3 and 6.2.3.4.1). When inserting components into an assembly, their BOM items are transferred to the assembly (see Section 6.2.3.4.2, method 1).
2. Automatic creation of BOM items that correspond to first-level components in an assembly (see Section 6.2.3.4.2, method 2) on the basis of the components' properties.

Both first and second methods form a set of BOM items in the assembly that reflect its contents. If necessary, you can create BOM items in the assembly that are not linked to components (see Section 6.2.3.4.2, method 4).

After attachment of the assembly to a BOM document (see Section 6.2.4.1), all BOM items are transferred to it from the assembly.



To correctly make BOM items in model, it is recommended to preliminary save it on the disk.

6.2.3.3. Creation of BOM items in part

As a rule a single BOM item is created in part, containing its name and designation and belonging to *Parts* section.



Similar to graphical documents, in order to create BOM items in the part you should call **BOM – Add item ...** command. It is not required to select any form-generating and structural elements before the command call.

Further actions (selection of section for placing the item, entering the textual part and setting additional parameters) are identical to those used to create BOM items in a drawing (see Section 6.2.3.1).

If current part (the root Design Tree element) was selected before the BOM item creation command calling, the *Designation*, *Name* columns and additional *Mass* column are filled in automatically, and current document-part is connected to the created BOM item.



Using automatic filling of the text portion of a BOM object is preferable over the manual input, since it helps avoid accidental errors.

6.2.3.4. Creation of BOM items in assembly

You may create internal and external items in assembly (please, see Section 6.1.3.2.2 on p. 1460).

6.2.3.4.1. External items

As a rule a single external BOM item is created in the assembly which contains its name and designation and belongs to *Assembly units*. This item is automatically transferred to the assembly in which the current one is included as subassembly (assembly unit).



To create external BOM item in the assembly call **BOM— Add item— External**.

Further actions (selection of section for placing the item, entering the textual part and setting additional parameters) are identical to those used to create BOM items in a drawing (see Section 6.2.3.1 on p. 1494).

If current part (the root Design Tree element) was selected before the BOM item creation command calling, the *Designation*, *Name* columns and additional *Mass* column are filled in automatically, and current assembly document is connected to the created BOM item.

6.2.3.4.2. Internal items

There are four methods of creating internal BOM items within an assembly. These methods can be combined at user's discretion.

1. Building BOM items when creating components.

The internal BOM items appear in the assembly on inclusion of parts containing BOM items and assemblies which, in turn, contain external BOM items. In other words, during insertion of components in the assembly, BOM items of parts and external BOM items of subassemblies become internal items of the main assembly.

If model documents have not been connected to "their" BOM items, this can be done while editing internal BOM items of the assembly (for more information on connection documents to BOM items see Section 6.2.2.6.3 on p. 1489. All the modifications applied to internal BOM items that have "arrived" to the assembly from components are transferred to the relevant component files.

Therefore, if you create BOM items in all the parts and subassemblies of the current assembly, the latter acquires all the items to be included into the BOM for this assembly. These items can be transferred to the BOM connected to the assembly.

This is the preferred method for creating BOM items corresponding to the relevant components in an assembly, since building BOM items while working directly with components reduces error probability.

2. Automatic Creation of BOM Items for components of the first-level assembly

If there is no BOM item in the component's file, its related internal BOM items is formed in the assembly based on properties (designation, name, mass) of the component. The object is formed automatically after the component's insertion into the assembly as long as there is the BOM's description in the assembly (see Section 6.1.3.4 on p. 1461). Automatically created object **is not transferred** to the component's file; it exists in the assembly only.

The style of the automatically created BOM item corresponds to the current BOM's style. Thus, after the new BOM description appears in the assembly and is selected as the current one, internal BOM items will be created in the BOM with the style that is included in the new description.



Note that the **Synchronize with the item's owner document** switch is disabled in the properties of the automatically created BOM item (see Section 6.2.2.6.1 on p. 1485); for this reason, the item is not transferred from the assembly to the component. When editing the BOM item, you can enable this switch. After this, the BOM item will appear in the component file.

This method makes it possible to avoid creation of BOM items in component files. Besides that, designations and names specified for components in an assembly are used in automatically created BOM items (see Section 5.1.4.1.2 on p. 1387). Generally, they can be different from designations and names in the component files. Thus, if necessary, a component can be quickly renamed in the assembly without making changes in its file.



This method of internal BOM items' forming is used during automatic creation of a BOM by assembly (see Section 6.2.3.8 on p. 1506).

3. Automatic Creation of BOM Items Corresponding to Components

Automatic creation of BOM items in the current assembly components is initiated by **BOM – Create BOM Items** command.

This command creates BOM items in all components that did not contain them. Parts are populated with regular BOM items whereas assemblies are populated with external BOM items. Designations and names are imported from the component properties to fill in a supplementary *Mass* column; the model documents are connected to the BOM items created within them.

See more details about the automatic BOM item creation command in Section 6.2.3.7.1 on p. 1504.

To use this method of creating BOM items, all assembly component files must contain their designations and names.

4. Building BOM Items in Assembly and Linking Them to Components and Bodies



In order to create in the current assembly an internal BOM item linked to some component or body built in this assembly, select the necessary item in the Design Tree and run the **BOM – Add Item – Internal** command.

Further actions (selection of section for placing the item, entering the textual part and setting additional parameters) are identical to those used to create BOM items in a drawing (see Section 6.2.3.1 on p. 1494).

Please note that the *Designation* and *Name* columns of the BOM items are automatically populated with the data entered when setting the component (body) properties whereas its current mass is entered to the *Mass* supplementary column. If a component has been selected, its file is automatically connected to the BOM item being created; after creation, this BOM item is transferred to the relevant model file (in case of subassembly, the newly created item becomes an external BOM item).

In some cases you may need to create an internal item in the assembly which cannot have a corresponding component or body. For example, some material is used while assembling the product. The material data must get into the BOM (i.e. the relevant internal item should be present in the assembly). However, it is not possible to build a model of this material. In this case, you must create an item in the assembly which is connected to neither an external file nor any body. To do this, use the **Add Item – Internal** command without highlighting any assembly items.

This method is the only one that enables you to create internal items in assemblies which may be either connected to bodies or unconnected neither to bodies nor components. However, be careful when using it: This method increases the probability of accidental creation of identical BOM items – including those for components. It is impossible to delete BOM items connected to components in an assembly. Therefore, to delete “irrelevant” items you should edit the component files.



The **Create BOM items** option controls forming internal BOM items in the assembly that correspond to components. By default, it is enabled.

If it is disabled, no BOM item will be created for the component (when using method 2) and it will not be transferred to internal assembly's BOM items (if any of the other methods is used) from the component file.

Therefore, if there is no BOM item in the assembly for a component, makes sure creation of the item is enabled. To do so, select the component in the Design Tree and call the **Component properties** command from the context menu. Check the status of the **Create BOM Items** option on the Property bar.

The **Create BOM Items** option is also available in the properties of library components.

6.2.3.4.3. BOOM objects from the Library of Standard Items

When inserting a model from the Library of Standard Items into an assembly, BOM object is created by default.

After selection of an object from the Library and specifying its position on the screen, a window for entry of the BOM item's text part will appear (similar to the one displayed during creation of the BOM item in the model).

After confirmation of item creation an internal BOM item will appear in the assembly. It will be placed in a certain section of the Library, for example, a fastener will be located in the *Standard items* section.

You may decide not to create the BOM object when inserting a model from the Library. Insertion of objects, BOM object creation are described in the Library's help system.



You can insert BOM objects from another application library in a way provided for by the library.

6.2.3.4.4. BOM items from model libraries

The model stored in the model library (*.I3d file), may contain the BOM items similar to the model stored in a separate file on the disk.

If to insert library component containing internal BOM items into the assembly respective internal BOM items will appear in the assembly. Work order with them doesn't differ from the work with the BOM items "came" into the assembly during components insertion from separate files.

This option may be used if your parts use standard parts and units.

1. Create 3D model of the standard part (unit).
2. Create external BOM item corresponding to the part (or unit).
3. Include the model into the model library.
4. Insert the model from the library into assemblies.

BOM items will be transferred together with the library component body.

6.2.3.5. Editing of the BOM items in the document



Editing of BOM items in drawings and models is made in slave mode (please, see Section 6.1.3.3 on p. 1461). This mode is available if a BOM description exists in the document. To call it use command **BOM – Edit items...**



You can proceed to editing the BOM item in the drawing using it's connected geometry. Select the necessary graphical objects and run **Edit BOM Item** command from the context menu.

BOM item changes made in assembly drawing may be transferred to the BOM item connected to it as the result of synchronization (please, see Section 6.2.4.2.1 on p. 1513).



Designation and name of the drawing affects the image of the BOM items containing code and name of the document. This influence is similar to the influence of BOM designation and name on items in it (please, see Section 6.2.2.10 on p. 1492).

6.2.3.5.1. Editing of the BOM item geometry contents

Methods of the BOM item geometry contents in the drawing (adding, replacing and deletion of geometry) are practically similar to the methods of editing of the geometry contents in the document-BOM (please, see Section 6.2.2.5.2 on p. 1483). The only difference is that you need to select the item with the contents being edited in the slave mode (i.e. directly within the drawing and not in a separate BOM document).

6.2.3.5.2. Review of the BOM item geometry

The order of review of the BOM items created in the drawing or assembly is actually similar to the order of item review in the document-BOM (please, see Section 6.2.2.5.3 on p. 1484). The difference is that BOM items contents of which is visible should be highlighted in slave mode.



When viewing geometry place the drawing (or assembly) window and the window of slave mode of BOM item editing as a mosaic.

6.2.3.5.3. Assembly BOM item synchronization with the owner document and component's properties

The document – container of the BOM item — is an assembly component to which an internal BOM item contained in the assembly is linked.

By default, retrieving designation and name from the source (respective options in the **Source** column are enabled, see Fig. 5.1.12 on p. 1386) is enabled in the component's properties immediately after insertion of the component into assembly. Note that:

- ▼ **If a BOM item was present in the component file,** this item is transferred to internal BOM items of the assembly. For this object, synchronization with the owner document is enabled, i.e. the **Synchronize with the item's owner document** switch is enabled in the **Settings** group on the Property bar. With such a setting, changes of a BOM item are transferred:



- ▼ from assembly to component — without any prerequisites; transfer takes place immediately after confirmation of the BOM item creation,
- ▼ from component to assembly — on condition that retrieving of designation and name from the source is enabled; transfer takes place when the component's file is saved.

Disabling the **Synchronize with the item's owner document** switch means that transfer of changes in either direction is prohibited.

▼ **If there was no BOM item in the component file,**

the object is automatically created in the assembly on the basis of the component's properties. The **Synchronize with the item's owner document** switch in the **Settings** group on the Property bar is not enabled for such item, since there is no BOM item in the owner document.

If you need to transfer an automatically created BOM item to the owner document, enable the **Synchronize with the item's owner document** switch.



Synchronization with the BOM item's owner document is disabled automatically, if a designation or name have been edited in the item.

You can either leave synchronization enabled or:

- ▼ transfer changes to the owner document; for this purpose, enable the **Synchronize with the item's owner document** switch.
- ▼ retrieve data from the owner document; for this purpose, enable the **Source text** switch.

Apart from synchronization of the BOM item with the owner document, the user can control the link between the BOM item, which corresponds to the assembly component, to the properties of the component. For this purpose, the **Synchronize with component's properties** switch is used.

- ▼ If the switch is enabled, changes of the component's designation or name will be transferred from BOM item to the component's properties and vice versa. As a result, designation (name) in the component's properties will always coincide with the designation (name) in the respective BOM item.
- ▼ If the switch is disabled, there is no transfer of data. As a result, designation (name) of the component in properties and in the BOM item may be different.



Synchronization of objects with owner documents and components' properties in the BOM document to which the assembly is linked is performed in the same way described above

6.2.3.6. Deletion of BOM item from the document

To delete BOM item from the drawing, go to (**BOM — Edit items ...**) slave mode, highlight the item to be deleted and press <Delete> or call **Editor— Delete** item command.



If **Delete geometry while deleting BOM item** option was enabled during BOM setting, respective graphic items (assembly component image) would also be deleted from the drawing. You can find more information on this in Section 6.2.6.1.1 on p. 1518.

To delete BOM item from the part go to slave mode, highlight the item to be deleted and press <Delete> or call **Editor – Delete** item command.

The same method is used for deleting external BOM objects from the assembly and those internal objects, which are either connected to bodies or not connected to bodies or components.

The internal BOM objects in an assembly which are linked to model documents cannot be deleted using the above method. You may exclude the BOM item belonging to the special component from the set of internal BOM items of the assembly. For this, do the following.



1. Select required component in the Design Tree.
2. From the context menu call the **Properties** command.
3. In Properties panel disable **Create BOM items** option.

The BOM object belonging to the selected component shall be excluded from internal BOM objects in the assembly. The item will not be deleted from the component itself. It just stops being available in the internal BOM items in the current assembly.

Excluding a BOM object belonging to some component from the internal BOM objects in one assembly in no way affects its presence among internal BOM objects in other assemblies, in which it enters as a component.

6.2.3.7. Automatic creation and deletion of the assembly BOM items

If during creation of assembly components their designations and names were entered BOM items in parts and subassemblies may be automatically created during working with the assembly.



Designation and name of the model are entered during its properties setting.

Inverse operation is also possible — automatic deletion of all BOM items available in the assembly and its components.

When setting automatic creation or deletion of BOM items you may enable creation or deletion of documents-BOM related to the assembly and its subassemblies. Description of the assembly— BOM relation, please see in Section 6.1.3.5.2 on p. 1463.

Commands of automatic creation and deletion of BOM items in the assembly are described in details below.

Note that the time required to automatically create or delete BOM items may be considerable, since opening, editing and saving of models takes place. Generally, the operation time depends on:

- ▼ contents of the assembly and complexity of its components,
- ▼ PC efficiency,
- ▼ location of the assembly files and components: on the disk of the same PC where KOMPAS-3D is running and the command of creation or deletion of BOM items is called (in this case the command is the fastest) or on network disks.

To save time, we recommend using these commands at the final step of working with the assembly, when all the necessary components have been included in it, rather than each time after adding a component.

6.2.3.7.1. Creation of BOM items



To create BOM items in parts and subassemblies of the current assembly call command **BOM—Create BOM items...**

The command is enabled if the current assembly is saved in a file on the disk.

The dialog shown in Fig.6.2.20 will be displayed. The control elements of this dialog are represented in Table 6.2.3.

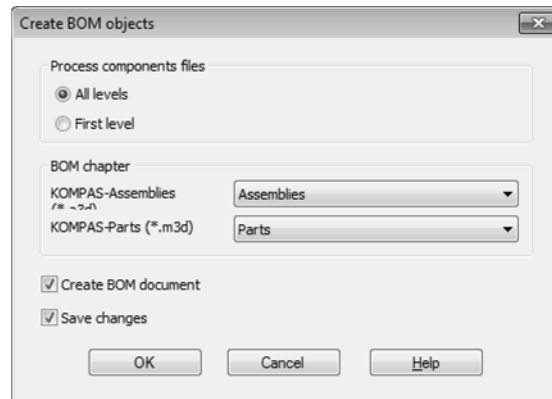


Fig. 6.2.20. BOM item generation dialog

Tab. 6.2.3. BOM item generation dialog

Element	Description
Proceed with Component Files	The group of elements enable to specify the scope of the command.
All levels	Enabling of this option means that: <ul style="list-style-type: none"> ▼ BOM items will be created in the current assembly parts and in the parts of subassemblies, ▼ external BOM items will be created in all subassemblies.
First level	Enabling of this option means that: <ul style="list-style-type: none"> ▼ BOM items will be created in the current assembly parts, ▼ External BOM items will be created in subassemblies of current assembly.

Tab. 6.2.3. BOM item generation dialog

Element	Description
BOM Section	Group of elements allowing to define to which sections BOM items should belong. KOMPAS-Assemblies (*.a3d) and KOMPAS-Parts (*.m3d) lists contain BOM item names of the current style. If the current style is not defined (i.e. if there is no BOM description) the style defined for new BOM is used. Select sections into which BOM items in parts files and external BOM items in assembly files will be placed. If BOM items should not be created in part files or in assembly files, select an empty line in the KOMPAS-Assemblies (*.a3d) or the KOMPAS-Parts (*.m3d) list.
Create BOM Document	Enable this option to automatically create and connect document-BOM (*.spw) to the current assembly. If Process component files group has All levels option enabled, the documents-BOM corresponding to these subassemblies will be automatically created and connected to subassemblies of all levels. Documents-BOM are named similar to the assemblies to which they are connected and are placed in the place similar to the assemblies location.
Save Changes	Enable this option to save the current assembly after creation of BOM items*.

* Component files are saved regardless of the option.

Having finished setting of BOM item creation, click the **OK** button of the dialog.

BOM items corresponding to the components will be created in those components which did not contain BOM items. Note that:

- ▼ data entered during setting of the component properties will be entered into *Designation* and *Name* columns,
- ▼ current component mass will be entered into the additional *Mass* column,
- ▼ component file will be connected to the created BOM item.



The **BOM – Create BOM Items** command cannot process bodies created in the assembly and library components. The same Library command may be applied to objects inserted from the Library of Standard Items.

6.2.3.7.2. Deletion of BOM items



To delete BOM items from the current assembly, its parts and subassemblies call command **BOM– delete BOM items...**

The command is available if the current assembly is stored in the file on the disk and contains BOM items.

The dialog shown in Fig.6.2.21 will be displayed. The control elements of this dialog are represented in Table 6.2.4.

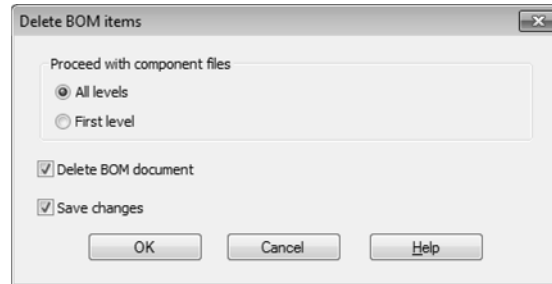


Fig. 6.2.21. BOM Item Deletion Dialog

Tab. 6.2.4. BOM Item Deletion Dialog

Element	Description
Proceed with Component Files	The group of elements enable to specify the scope of the command.
All levels	Enabling of this option means that all BOM items will be deleted from the current assembly.
First level	Enabling of this option means that: <ul style="list-style-type: none"> ▼ all BOM items will be deleted from the current assembly, ▼ all external BOM items will be deleted from subassemblies, ▼ external and internal BOM items not connected with components (for example in <i>Materials</i>) section will be deleted from the current assembly.
Delete BOM Document	Turn on this option to delete the BOM document (*.spw) linked to the current assembly. If the All Levels option is on in the Process Components Files group, the BOM documents linked to the subassemblies are deleted automatically.
Save Changes	Turn this option on to save the current assembly and all its components after the BOM items have been deleted.

When done customizing the BOM item deletion, click **OK** in the dialog.

6.2.3.8. Creation of BOM by assembly

You can automatically create a BOM with items that correspond to first-level components included in the BOM.



For this purpose, open the required assembly and call the **BOM — Create BOM by assembly** command. The command is not available if the assembly is not saved in a file on disk or is being edited in the context of a different assembly.

After calling the command, the filled BOM will appear.

If any BOM components had no BOM items, they will be created automatically on the basis of these components' properties.



The new BOM document is not saved to file automatically. The save command should be called manually. Only after this, the BOM will be synchronized with the assembly; for example, the BOM will be changed after components' properties are edited in the assembly or components are added/deleted. Synchronization is performed automatically when the assembly is saved.

The new BOM has the style that was assigned during new documents' setup. If the assembly has no versions, the BOM style is defined by the default setting (see Section 9.2.5.1.1 on p. 1755); style of the BOM by assembly with versions depends on the number of versions (see Section 9.2.5.2.1 on p. 1758).



An assembly cannot be simultaneously attached to several BOMs with the same style. Therefore, the new BOM is created after execution of the **Create BOM by assembly** command if the assembly is not linked to a BOM with the same style. Otherwise, the BOM will be loaded (or enabled, if already loaded) to which the assembly is attached.

When working with the command for creation of BOM by assembly, the following operations are performed automatically:

- ▼ Creation of internal BOM items in assembly for components in the files of which these items did not exist (see Sections 6.1.3.2.2 on p. 1460 and 6.2.3.4.2, p. 2 on p. 1498 for more information about internal BOM items). You can view or edit internal BOM items of the assembly in the slave mode (see Section 6.1.3.3 on p. 1461).
- ▼ Creation of a BOM document and its attachment to assembly (see Section 6.1.3.5.2 on p. 1463 for more information about linking assembly to BOM).

When using the command for creation of BOM by assembly, note the following:

- ▼ The command does not create any BOM items for bodies or components inserted from applied libraries.
- ▼ If the **Create BOM items** option in the component's properties is disabled, the internal BOM item that corresponds to the component will not be created in the assembly.

6.2.4. Connection of BOM with drawings and models

6.2.4.1. Mutual connection of documents

Connection between the BOM and the assembly drawing or between the BOM and the model-assembly may be set in two ways:

- ▼ by connection of BOM to the drawing or assembly (please, see Section 6.2.4.1.1),

- ▼ by connection of drawing or assembly to BOM (please, see Section 6.2.4.1.2).

Properties of connection between assembly or drawing and BOM do not depend on the order of documents connection to each other. In other words, having BOM and assembly connected to each other (or BOM and drawing) it is impossible to define whether the assembly (drawing) has been connected to BOM or the BOM has been connected to assembly (drawing).

6.2.4.1.1. Connection of BOM to the current assembly or drawing

Connection of BOM to assembly or drawing may be made at any stage of working with this document. The only condition of connection is availability of named file of the current assembly or drawing on the disk. I.e. you can't connect the BOM to the document which was never saved from the moment of creation and thus has no name.

It is possible to connect existing BOM (including those containing items) to assembly or drawing or to create a new BOM during connection.



1. To connect BOM to the current document call command **BOM – Control BOM descriptions**. BOM description control dialog will be displayed (Fig.6.2.22 or 6.2.24). Please see detailed description in Section 6.1.3.4 on p. 1461.



During automatic creation of BOM items in assembly (please, see Section 6.2.3.7.1 on p. 1504) it is possible to automatically create a document-BOM and connect it to the current assembly.

If the current document has BOM items...

Assembly or drawing already has description of BOM. This description contains no BOM file name. Only BOM style is defined in it (Fig. 6.2.22).

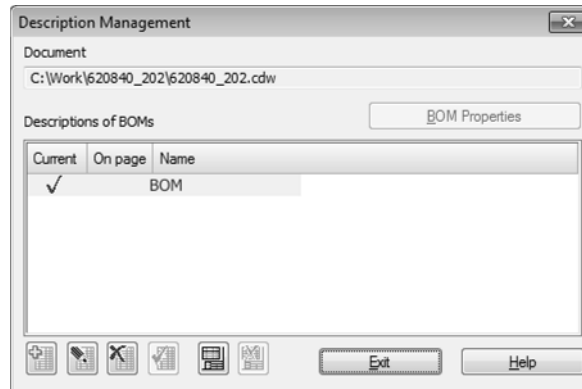


Fig. 6.2.22. BOM description if BOM items are available in document



2. In dialog of BOM descriptions control select existing description and click the **Edit description** button.
3. In appeared dialog of BOM description (Fig. 6.2.23) define file name of the connected BOM. For this click the **Select** button in the group of options **BOM file name** and in the appeared dialog define BOM file name.

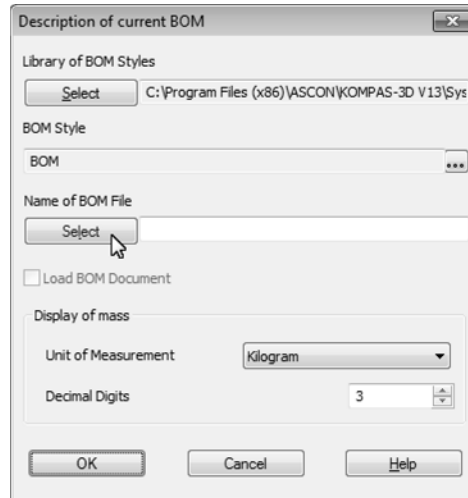


Fig. 6.2.23. Description editing

BOM description dialog will be closed. Full name of the BOM connected to document will be added to BOM style name in the dialog of BOM description control.



Indicated BOM should have the style similar to that of IBOM existing in the assembly or drawing.

If you enter name of not existing BOM in file opening dialog the system will create the BOM with indicated name. Style of this BOM will correspond to BOM items in the drawing.



The BOM style indicated in the dialog can't be changed since BOM items already exist in the document created according to this style.

If there is no BOM items in the current document...

List of BOM descriptions in BOM description control dialog is empty (Fig. 6.2.24).

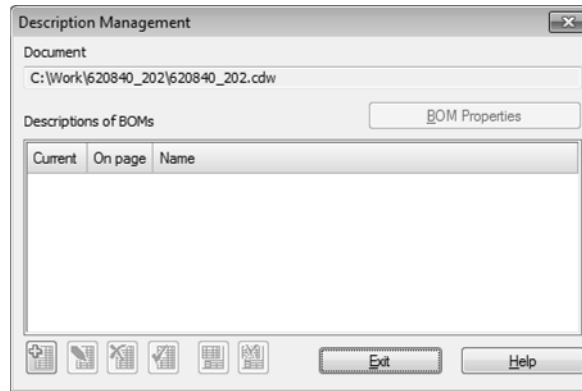


Fig. 6.2.24. BOM description if there is no BOM items available in document



4. To add a description to it in the dialog click the **Add description** button.

BOM description dialog will be displayed (Fig. 6.2.23).

5. Specify BOM file name and its style in it.

BOM file name unambiguously defines its style.

- ▼ If the BOM which should be connected already exists just select its style: click the **Select** button and define BOM file in the appeared dialog. After this style name library and name of the selected BOM style will appear in respective fields.
- ▼ If the BOM which should be connected doesn't exist, define the style it should have: first select style library and then— style name from it. After this click the **Select** button in **BOM file name** group and in the appeared file opening dialog enter name of not existing BOM. The system will create BOM with specified name and selected style.

After fulfillment of points 1, 2 and 3¹, description containing name of the document-BOM will appear in the dialog of descriptions control (Fig. 6.2.25).

1. Points 2 and 3 are made once, variant of fulfillment is selected depending on the availability of BOM items in the current document.

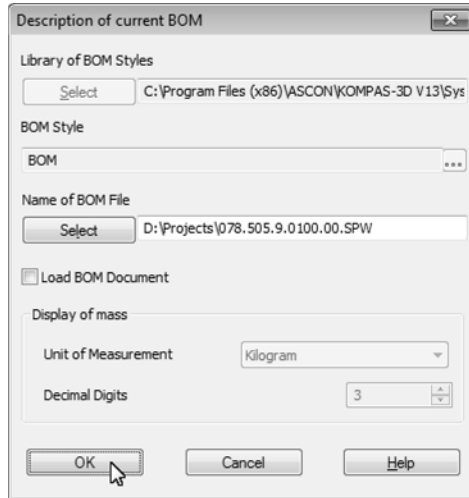


Fig. 6.2.25. Description editing

6. In description control dialog select the name of the BOM which is required to connect to the drawing or assembly and click **OK**.

BOM will be connected to the document.



If you enable **Load BOM document** option in the description dialog and then click the **OK** button to exit the dialog, connected BOM will be opened in KOMPAS-3D.

7. To exit the description control dialog click the **Exit** button.

6.2.4.1.2. Connection of assembly or drawing to the current BOM



You may connect any number of assemblies and drawings to BOM. Usually 3D model of assembly or (and) pages of assembly drawing are connected to BOM.

1. To connect assembly or drawing to the current BOM, call **Service— Assembly control** command.

Assembly control dialog will be displayed (Fig. 6.2.26). It displays names of the documents connected to the current BOM.

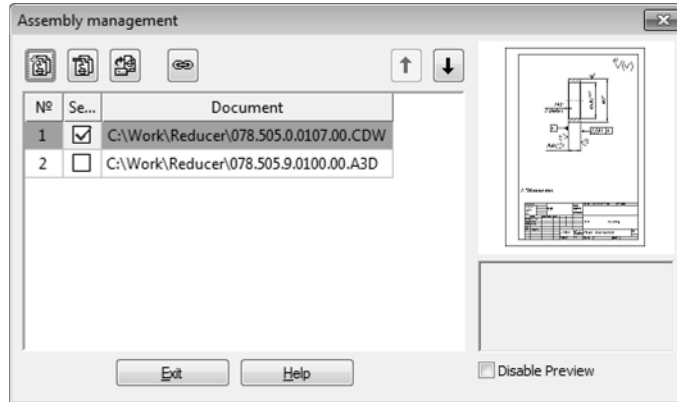


Fig. 6.2.26. Assembly Management Dialog



2. To connect assembly or drawing to the BOM in the dialog click the **Connect document** button.

3. In the appeared file selection dialog specify file of the document.



4. To enable transfer of the designation and name from the first connected document to main text of the BOM click the **Fill in title block** button.

5. To mark the documents into which designation and name from the main BOM text should be passed, enable options in respective cells of the column **Transfer changes to document**.



You may quickly open any of the connected documents. For this purpose, select it in the list and click the **Edit document** button.

6. To close assembly control dialog click the **Exit** button.

Thus **Fill in main text** button controls data transfer from the connected document to BOM. While it is pressed connection between the BOM and the first connected document is kept. If the other document will turn out to be in the first place in the list the designation and name from this document will be transferred into the BOM. The order of document sequence in the assembly control dialog may be changed with **Move document up** and **Move document down** keys.



Document sequence order in the list is also considered in area calculation.

Upon releasing the **Fill in Title Block** button, the data transfer to the BOM's title block will stop. The product's current designation and name change neither in the document nor in the BOM.

The **Send Changes to Document** column controls product designation and name transfer from the BOM into the connected documents. If to disable option in this column at any document, transfer of the part designation and name from the BOM into this document will be stopped. The product's current designation and name change neither in the document nor in the BOM.

Therefore, in order to set two-sided connection between the BOM and connected document in control dialog click the **Fill in main text** button, place required document in the first place in

the list of connected documents and enter mark for it in column **Transfer changes into document** (as it was done, for example, for *Conveyor.cdw* document in Figure 6.2.26).

6.2.4.2. Data synchronization

Usually BOM items are created and edited in one of the documents— connected to each other or in assembly drawing (or model) or in BOM. The result is that items in the documents connected to each other differ.

For correct processing mutually connected documents should have similar BOM items. To achieve this make synchronization— transfer actual items to the connected document (please, see also Section 6.1.3.6 on p. 1464).

Besides, part designation and name in the BOM and connected documents should be the same.

6.2.4.2.1. Transfer of items from drawing or assembly into the BOM



To transfer BOM items from the current drawing or assembly into the BOM connected to this document, call **BOM – Synchronize data** command.



Connected BOM should not necessarily be opened in the moment of command call.

Through the command items from the drawing or internal items from the assembly should be transferred to the BOM. The system will display a message about the BOM changing.

During saving synchronization of the drawing with its BOM is made automatically.

If during setting of BOM **Connection of assembly or drawing with BOM** option is disabled (please, see Section 6.3.2.1 on p. 1547), no synchronization will be made.

6.2.4.2.2. Transfer of items from the BOM into the drawing or assembly



To transfer items from the current BOM to the documents connected to it, call **Service— Synchronize data** command.



Connected document should not necessarily be opened in the moment of command call.

Through the command items from the BOM will be transferred into the drawing or assembly. If during connection of documents to BOM items **Transfer changes into documents** option is enabled, data from the item columns will be transferred to the documents connected to items (for example, cells of main text of parts drawings into respective cells). The system will display a message about changing of BOM and documents connected to items.

Not all BOM items but only those created (and thereafter transferred to BOM) in the drawing are transferred to this drawing or have in their contents geometry from this drawing. This rule exists because BOM may be connected to several pages of assembly drawing and transfer of all BOM items in every connected drawing will result in big number of similar items. To prevent such complex situation the item of the BOM will be transferred only to the drawing where the image corresponding to it is located.

Transfer of items from the BOM to connected drawings is also made every time during BOM saving. In this case synchronization occurs automatically and there is no need to call respective command.

6.2.4.2.3. Data transfer between the BOM main text and connected documents

Synchronization of parts designation and name is made automatically during saving of any from connected documents. It is not required to call special command. Data are transferred as follows.

Full content of columns of main text *Designation* and *Name* is taken from the document (if the document is BOM or drawing) or *Designation* and *Name* fields (if the document is an assembly model).

If the trailing characters of the resulting designation make up one of the standard codes, then the designation is passed to the linked document, stripped off those characters. Otherwise the designation will be transferred “as is”. The product name is processed similarly.

For example, Figure 6.2.27 shows main texts of drawings connected to BOM and Figure 6.2.28— main texts of BOM filled with the data received from these drawings. Since the drawing on Figure 6.2.27, a) has standard code and name, main text of the BOM of Figure 6.2.28, a) is filled in correctly. Drawing in Figure 6.2.27, b) has non-standard code, that is why BOM designation in Figure 6.2.28, b) is not correct.



List of standard codes and names is defined by code and name files — *graphic.kds*. By default it is stored in \Sys subfolder of the main system folder. File format— is a text one that is why if required *graphic.kds* file may be opened and edited in any text editor.

					KSKV.12.020SB			
						<i>Lit.</i>	<i>Mass</i>	<i>Scale</i>
<i>Issd. Sheet</i>	<i>Nº</i>	<i>Sign.</i>	<i>Date</i>	<i>Cantilever Assembly drawing</i>		6,146	1:2	
<i>Resp. Eng.</i>	<i>Ivanov</i>		24.01.00					
<i>Check. Per.</i>	<i>Ivanchenko</i>		26.01.00					
<i>Mfg. Eng.</i>								
<i>Qual. Eng.</i>								
<i>Appr. Per.</i>	<i>Polikarpov</i>		01.02.00		<i>Sheet</i>	<i>Sheets</i>	1	
					Konbers LTD			

a)

Fig. 6.2.27. Main texts of drawings

					KSKV.12.020SBB			
					Cantilever Assembly drawing	<i>Lit.</i>	<i>Mass</i>	<i>Scale</i>
<i>Bord.</i>	<i>Sheet</i>	<i>№</i>	<i>Sign.</i>	<i>Date</i>			6,146	12
<i>Resp. Eng.</i>	Ivanov			24.01.00				
<i>Check. Per.</i>	Ivanchenko			26.01.00				
<i>Mfg. Eng.</i>						<i>Sheet</i>	<i>Sheets</i>	1
<i>Qual. Eng.</i>					Konbers LTD			
<i>Appr. Per.</i>	Polikarpov			01.02.00				

b)

Fig. 6.2.27. Main texts of drawings

					KSKV.12.020			
	<i>Sheet</i>	<i>№</i>	<i>Sign.</i>	<i>Date</i>	Cantilever	<i>Lit.</i>	<i>Sheet</i>	<i>Sheets</i>
<i>Resp. Eng.</i>	Ivanov			25.01.00				1
<i>Check. Per.</i>	Ivanchenko			27.01.00				
<i>Qual. Eng.</i>						Konbers LTD		
<i>Appr. Per.</i>	Polikarpov			01.02.00				

a)

					KSKV.12.020SBB			
	<i>Sheet</i>	<i>№</i>	<i>Sign.</i>	<i>Date</i>	Cantilever	<i>Lit.</i>	<i>Sheet</i>	<i>Sheets</i>
<i>Resp. Eng.</i>	Ivanov			25.01.00				1
<i>Check. Per.</i>	Ivanchenko			27.01.00				
<i>Qual. Eng.</i>						Konbers LTD		
<i>Appr. Per.</i>	Polikarpov			01.02.00				

b)

Fig. 6.2.28. Main texts of BOM

6.2.5. Standard procedures of BOM creation

Methods submitted in Sections 6.2.2 – 6.2.4 allow creating BOM using different methods. All these methods may be randomly combined to get BOM of required view.

But several standard BOM creation procedures exist among possible combinations of ways and methods. They are described in this section. These procedures are optimal from the point of view of the BOM creation speed and generation of correct set of design documents for the part.

6.2.5.1. BOM Related to Assembly Drawing

1. Switch to slave mode with BOM items of the drawing (please, see Section 6.2.3.1 on p. 1494). It is desirable for them to have geometry in its contents (at least, leaders).
2. Switch to slave mode with BOM items of the drawing (please, see Section 6.2.3.5 on p. 1501). Be sure that the BOM table has all items created by you available.
3. Connect the document-BOM to the drawing (please, see Section 6.2.4.1.1 on p. 1508).
4. Transfer items from drawing to BOM (please, see Section 6.2.4.2.1 on p. 1513).
5. Save your BOM.



If during creation of BOM items you connect drawings to them, a copy of the geometry (“the base” of the geometry) from the assembly drawing may be made (please, see Section 6.2.6.4 on p. 1521).

6.2.5.2. BOM Connected to Assembly Model

1. Create and save the component files of the would-be assembly (parts and subassemblies).
2. In every parts file generate a BOM item corresponding to it (please, see Section 6.2.3.3 on p. 1497), placing it in *Parts* section.
3. In every subassembly part generate an external BOM item corresponding to it (please, see Section 6.2.3.4.1 on p. 1497), placing it in *Assembly parts*.
4. Create an assembly document and add to it all required components.
5. If necessary, add objects from the Library of Standard Items to the assembly (see Section 6.2.3.4.3 on p. 1500).
6. If some assembly components can be rebuilt only in its context, create these components. When in the context editing of a component, create BOM objects: for the part — standard item, for the subassembly — external one.
7. Switch to slave mode of operation with internal items of assembly BOM (please, see Section 6.2.3.5 on p. 1501). Be sure that *Assembly units* section contains external BOM items which belong to subassemblies, *Parts* section — BOM items which belong to parts, *Standard parts* — BOM items which belong to library elements, and values in *Quantity* column correspond to the number of component insertions into the assembly.
8. If required, create internal BOM items in the assembly (please, see Section 6.2.3.4.2 on p. 1498) — for example, in *Materials* section.
9. Save the created assembly file.
10. Connect document-BOM to the assembly (please, see Section 6.2.4.1.1 on p. 1508).
11. Transfer items from the assembly to BOM (please, see Section 6.2.4.2.1 on p. 1513).
12. Save your BOM.
13. If necessary, insert positioning leaders into the assembly (see Section 6.2.6.17 on p. 1536).



As an alternative to the procedure described above, automatic creation of BOM using the **Create BOM items...** command can be used. (see Section 6.2.3.7.1 on p. 1504) or **Create BOM by assembly** (see Section 6.2.3.8 on p. 1506).

6.2.5.3. Complete set of associative documents

1. Generate internal BOM items in the assembly (please, see Section 6.2.3.4.2 on p. 1498).
2. Create an associative drawing of the assembly. Internal BOM items will be automatically transferred in it from the assembly.
3. Mark position leaders in the drawing. Include respective BOM items in the contents.
4. Connect the document-BOM to the assembly and (or) associative assembly drawing (please, see Section 6.2.4.1.1 on p. 1508).
5. Transfer items from the assembly or drawing into BOM (please, see Section 6.2.4.2.1 on p. 1513).

6.2.5.4. The BOM not related to the other documents

1. Create a document-BOM (please, see Section 6.2.2.1 on p. 1470).
2. Create BOM items in it and fill in their text part (please, see Sections 6.2.2.3 on p. 1471, 6.2.2.4 on p. 1476).



Use this way only for quick transfer of already issued “paper” BOM to electronic form provided that other documents for the part (assembly drawing, model detail design) are not available in electronic form.

6.2.6. Advanced capabilities

BOM design system has service features which are not strictly required to be used during BOM generation. They make some special additional functions.

Some additional methods of work with BOM and setting options described in this section may turn out to be useful for you. Of course, advices to use them are not to be taken as an instruction to use them, it's just an example of the using.

6.2.6.1. Operations with items containing geometry

The following automatic actions will be made if BOM items have graphical items in the contents:

- ▼ deletion of geometry with deletion of BOM item (please, see Section 6.2.6.1.1),
- ▼ deletion of BOM item with deletion of geometry (please, see Section 6.2.6.1.2),
- ▼ copying of BOM item during copying geometry (please, see Section 6.2.6.1.3).

Enabling and disabling of these service features is made in **Settings** Tab of BOM setting dialog (Fig. 6.2.29). To call the dialog use **Format— BOM setting...** command.

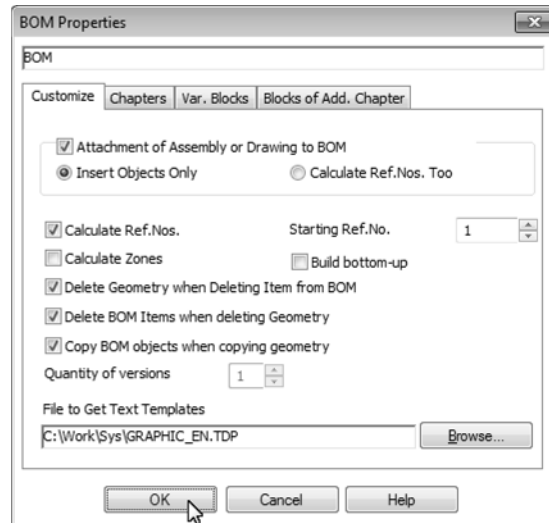


Fig. 6.2.29. BOM properties dialog



If you are working in the slave mode of editing BOM objects, and no BOM document is connected to the current drawing, then the BOM setup does not occur and the setup command is unavailable.

Using of listed features is possible both for working in document-BOM and in drawing in slave mode.

6.2.6.1.1. Deletion of geometry with deletion of the BOM item

Delete geometry with deletion of BOM item option (please, see Fig. 6.2.29) controls automatic deletion of the geometry of deleted BOM item from the drawing.

If the option is enabled graphical items included in BOM item contents will be deleted during BOM item deletion (even if this drawing is closed at the moment of BOM item deletion).



Be extremely careful when using the geometry deletion option when deleting a BOM object, since it is not always possible to restore objects deleted by this method.

6.2.6.1.2. Deletion of BOM items during deletion of geometry

Delete BOM items with geometry deletion option (please, see Fig.6.2.29) controls automatic deletion of BOM items with deletion of graphical items from the drawing included into contents of these BOM items.

If this option is enabled the dialog shown in page 6.2.30 is displayed during deletion of geometry objects and leaders included into contents of BOM items.

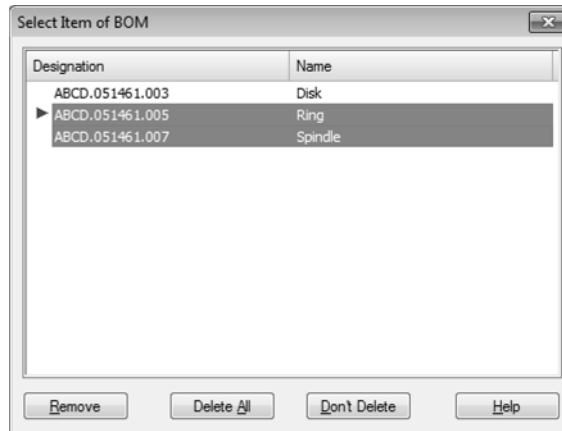


Fig. 6.2.30. BOM Item Selection Dialog

The dialog window lists designations and names of all BOM items, contents of which is fully deleted with deletion of selected geometry items. In this window, you can:

- ▼ delete all items containing deleted geometry by clicking the **Delete all** button,
- ▼ delete only some items highlighting them and clicking the **Delete** button,
- ▼ refuse to delete items by clicking the **Don't delete** button.

6.2.6.1.3. Copying of BOM items with coping of geometry

Copying of BOM items with copying of geometry option (please, see Fig.6.2.29) controls automatic copying of BOM items during copying of graphical objects in the drawing included into contents of these BOM items.

If this option is enabled a copy of BOM item is automatically created during copying in the drawing of:

- ▼ all geometric objects and leaders included in this object,
- ▼ all geometric objects included in this object,
- ▼ at least one leader included in this object.



If the BOM item geometry is located in different views of the drawing, this BOM item is also copied with copying of the part of the geometry fully contained in one view.

For copying BOM items it doesn't make sense which way you use to copy geometry — using mouse or editing commands.

6.2.6.2. Calculation of the sum of values in BOM columns

When setting some columns of BOM blank form and additional columns containing numeric data these columns may have data addition provided.



To add up numeric data in columns call command **Service— Add up values in columns**.

In the displayed dialog (Fig.6.2.31) you will see adding results. That will be performed according to the settings of the current BOM style. The values from columns with enabled adding option will be added. If the BOM is of group type the adding result will be provided for every variant.

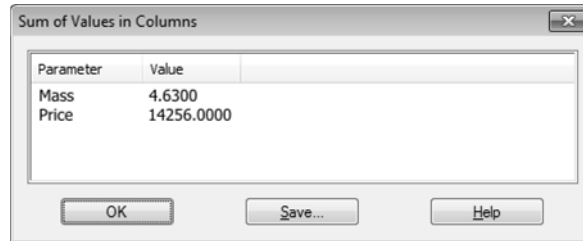


Fig. 6.2.31. Dialog with adding results

In this way you can calculate the mass or cost of a product being specified (that is, provided that you entered the mass or cost of all BOM objects).

You can save the results of summing up into a separate text file. For this click the **Write...** button in the dialog with results and enter file name.

6.2.6.3. Processing of BOM numerical columns

Sometimes when working with BOM it is required to make arithmetic operations with values in BOM columns (for example, to automatically fill in *Cost* column if the price and quantity of parts are known).

To do this use KOMPAS application library (*komlib.rtw* file). It is included into KOMPAS-3D delivery set and doesn't require separate key license hardware protection.

To calculate values in columns connect KOMPAS system library and select **Processing of numeric columns** in its **Operations with BOM** section.

Dialog of numeric column processing will be displayed (Fig.6.2.32, Table). 6.2.5).

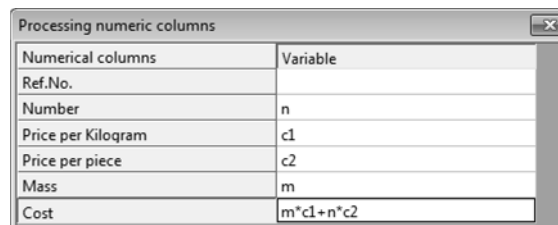


Fig. 6.2.32. Dialog of numeric column processing

Tab. 6.2.5. Controls of the dialog of numeric column processing

Option	Description
Numeric columns	Columns of active BOM (columns with <i>Integral</i> or <i>Real-valued</i> data type) are listed in this column. Both main and additional columns are included in the list.
Variables	In this column enter or edit <ul style="list-style-type: none"> ▼ names of variables which will be assigned to values of respective columns and ▼ equations to calculate data in other columns.

After entering of variables and equations click the **OK** button in the dialog. Calculated values will be entered in columns.



To recalculate values in columns (for example, after changing of initial data) it is required to repeat calling of **Processing of numeric column of BOM** command.

6.2.6.4. Creation of the base drawing based on the BOM item geometry

By the moment of BOM creation working drawings of parts and units may be not ready yet. BOM design system allows creating “base parts” of working drawings of parts and assembly drawings of assembly units and complexes.

The base part may be created if the drawing designed to image BOM item (for example, part) has not exist yet.

During connection of document (please, see section 6.2.2.6.3 on p. 1489) to BOM item enter non-existing drawing file name in the standard file selection dialog. The system will create a drawing with the name you specified and will put in it all graphical elements belonging to the contents of a BOM object (except a position reference number leader). You will get a template of a part working drawing or an assembly unit drawing.

You have to manually enter dimensions in it and add geometry, and some of the columns of the main text (for example, *Designation* and *Name*) will be automatically added. In the future, when editing text in those entities, the changes will be passed into a BOM.



Use this feature only if you do not have a 3D model of the assembly component. If such model is available create its associative drawing, not the drawing based on the geometry from assembly drawing.

6.2.6.5. Breaking of BOM on pages

Automatic breaking on pages takes place if there is no enough room for BOM sections on one page. In normal mode of BOM displaying upper and lower page borders are shown as horizontal dashed lines.

Sometimes during automatic BOM breaking in pages section header appears on one page and items of this section on the other page or multi-line BOM item is displayed on several pages. In such cases you may manually break the BOM on pages to have section header or BOM item on a new page.



To place section header or BOM item at the beginning of a new page select this header or item and activate switch **Place on new page**, located on Properties panel.

6.2.6.6. Creation of reserved lines in the middle of the BOM section

Usually reserved lines are located at the end of sections. Do the following if reserved line should be located in the middle of the section.

1. Using an additional BOM item create an empty line in the place of the section where reserved line should be located.



2. Select created line and activate switch **Item position increases**, located in Properties panel. Activation of this switch means that the item position number should be more by one than the number of the preceding item. If the option is disabled the number of the position matches the number of the previous item position.

6.2.6.7. Enabling and disabling of item displaying in table of BOM

Show item in the table option is disabled in settings during creation of some BOM items (please, see detailed description in section 6.2.2.6.1 on p. 1485).

To display hidden items in BOM table do the following.



1. Temporarily enable to modes of displaying all hidden BOM objects. For this use command **View— Show all items**. After calling the command, the respective button stays pushed (thus, it indicates the mode of displaying all objects).

After all BOM objects become visible as a result of this command, you can turn on the option of displaying in the table each particular object.



2. Select an item in the table and activate switch **Show item in table**, located in Properties panel. An object that was set up in this way will be displayed in the BOM table in any mode.
3. To disable the mode of all items showing, call **Show all items** command again. In this case the respective button will be released.



To disable an object display in the table, you do not have to go into the mode of displaying all objects. Just disable **Show item in table** switch for this item.

6.2.6.8. BOM saving in the other formats

Sometimes it is required to hand the BOM over to the other working places or enterprises. It may occur that the destination place has no KOMPAS-3D or BOM design system installed and that is why it is impossible to work with KOMPAS-3D BOM.

To make review and printing out of BOM in such working places possible you may export the BOM (*.spw file) to files of other formats.

6.2.6.8.1. Exporting into fragment

You may export the current BOM into KOMPAS-3D fragment file (*.frw). For this, do the following.

1. Connect KOMPAS application library (*komblib.rtw* file).
2. From its **Operations with BOM** section call **Convert BOM into fragment** command.
If BOM was active in KOMPAS-3D during command call, this (current) BOM will be exported. The system will display a standard file opening dialog if not the BOM was the current document at the moment of command call. Define required BOM in it.
Composition dialog will be displayed if exported BOM has more than one page (Fig.6.2.33).
3. Define in it how to place BOM pages in the fragment field (Table 6.2.6).

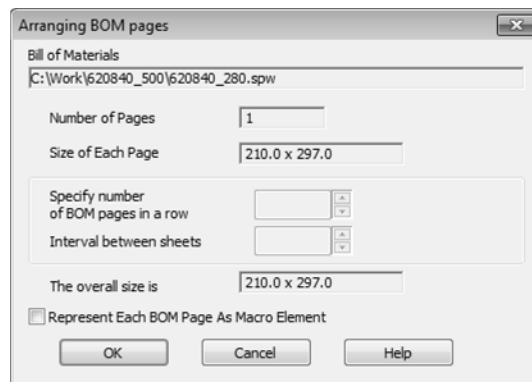


Fig. 6.2.33. Dialog of BOM pages composition

Tab. 6.2.6. Elements of composition dialog control

Interface item	Description
Bill of Materials (BOM)	Reference field where full file name of saved BOM is indicated.
Quantity of pages	Reference field where number of pages of saved BOM is indicated.
Dimension of every page	Reference field where dimension of pages of saved BOM is indicated (in mm).

Tab. 6.2.6. Elements of composition dialog control

Interface item	Description
Quantity of pages BOM Table in a row	You may enter in this field quantity of pages which should be place in horizontal row. For constant general quantity of pages indicated quantity simultaneously means quantity of rows where all BOM pages will be placed.
Spacing between pages	You may enter distance (in mm) in this field which should be remained between external frames of the BOM neighboring pages.
General dimension	This reference field indicates general dimension of the exported BOM image for selected composition. To change the overall extents (for example, to fit a BOM on a sheet of paper of a specified size), edit the value of the interval between the sheets and their position in the rows.
Every page of BOM to write as macroelement	If this option is enabled lines and texts containing every list of BOM create macroelement in the resulting file.

4. After defining of all composition parameters click **OK** to begin export.

The system will create a new fragment and place BOM page image in it. This image will consist of KOMPAS-3D graphical objects — lines and texts. If respective option is enabled they will be collected into macroelements.

All information related to the initial BOM (items, settings etc.) will be lost in the fragment. Only text parts of items (as separate texts in cells), BOM table (as lines) and main text (as lines and separate texts in cells) will be remained in it.

In the resulting fragment all editing features of KOMPAS-3D graphical documents editing will be available.



No relation between new fragment and initial BOM will occur. That is why you have not to expect that changes made in the BOM will be reflected in the fragment made from it.

6.2.6.8.2. Export into DXF, DWG and IGES formats

You may export the BOM into DXF, DWG and IGES formats. For this, do the following.

1. Open the BOM which should be exported.
2. Call **File— Save as** command and in **File type** field of appeared dialog select required file extension (*dxf*, *dwg* or *igs*).
3. Click the **Save** button.

Parameters button will appear in case of selection of **DXF** line in **File type** list or **DWG** in the dialog. It serves to call the dialog to set up document export in the selected format. Here, you can specify the format's version number, set up export of text and special symbols. For a multiple BOM, you can set parameters of pages' location in the resulting document's area (num-

ber of pages in a row and interval between them). After finishing the setup close the dialog by clicking the **OK** button, and then click the **Save** button of the file saving dialog.

Export setting for IGES format is impossible.

The system will create a file of required format and place BOM pages image in it. The image (similar to export into fragment) will consist of lines and texts.

Although files received during export lose properties of KOMPAS-3D BOM (document filling order, connection with assembly drawing, auto sorting feature and position marking etc.) and can't be further edited using means of BOM design System they are essential for the review and printing of BOM in those working places where it is impossible to work with KOMPAS-3D BOM files.

6.2.6.8.3. Export into DB format

To transfer data into automated system of production control you may need to convert BOM into dBase III DOS, dBase III Windows or Excel file formats.

To export BOM into DB table do the following.

1. Connect KOMPAS application library (*komlib.rtw* file).
2. From its **Operations with BOM** section call **BOM export** command.

After command calling dialog of BOM export parameters setting will be displayed (Fig.6.2.34–6.2.37).

3. Define export parameters in it (Tables 6.2.7 – 6.2.10).
4. After setting of BOM export parameters click the **Start record** button.



The more quick way to write the current BOM into Excel format is to call **File— Save as...** command. In the appeared dialog enter file name, select in **File type** list **Excel files (*.xls)** line and click the **Save** button.

The BOM will be saved in the default configuration in the Excel file format.

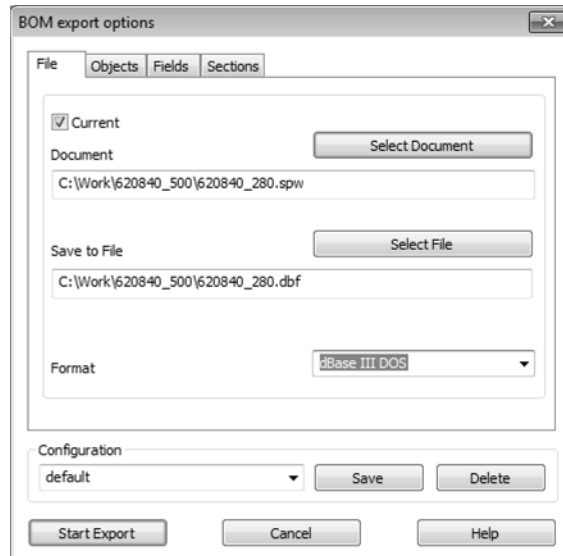


Fig. 6.2.34. Dialog of the BOM export parameters setting.
File Tab

Tab. 6.2.7. Dialog of the BOM export parameters setting. **File** Tab

Interface item	Description
Current	Enabled option means that the current document will be exported.
Select document	This button allows selecting BOM for export.
Document	Name of exported BOM is displayed in this field.
Save to File	Name of created file is displayed in this field.
Select File	This button allows defining the path to the created file.
Format	The list serves for selection of the type of file being created.

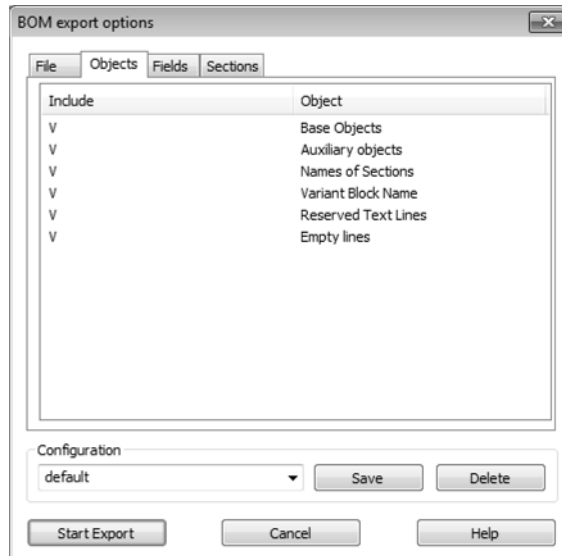


Fig. 6.2.35. Dialog of the BOM export parameters setting. Items **Tab**

Tab. 6.2.8. Dialog of the BOM export parameters setting. Items **Tab**

Interface item	Description
Include objects	<p>This window displays types of lines existing in the active BOM:</p> <ul style="list-style-type: none"> ▼ base items, ▼ additional items[*], ▼ sections headers[*], ▼ reserved text lines[*], ▼ empty lines[*], ▼ headers of variant blocks[*]. <p>Those which will be transferred to BD table are marked with check mark in Include column. To exclude lines of definite type from the exported ones left click the check mark.</p>

* Records with empty lines will be created during export of reserved and empty lines into DB table, records with small information scope (for example, with "Parts" text in *Name*) field are created during export of section headers and variant blocks (and sometimes additional objects). As a rule such records are useless during processing of DB by production control systems (or other expert or analytical programs). That is why export of respective lines is disabled. Special fields are included into the records to protect them from losing information about the section to which they belong (please, see Table 6.2.9).

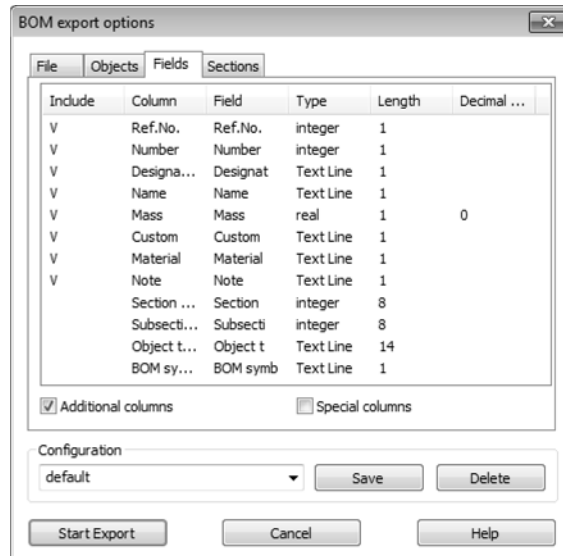


Fig. 6.2.36. Dialog of the BOM export parameters setting.
Fields tab

Tab. 6.2.9. Dialog of the BOM export parameters setting. **Fields** tab

Interface item	Description
Include columns	In this window columns of active BOM and field corresponding to them with current settings are listed in the table form. Fields which will be transferred to the BD table are marked with check mark in Include column. To change parameters (name, length, etc.) of any BOM field in a new document or to exclude the field from exported ones double click the respective line of the table.
Additional columns	This option controls export of all additional columns. If you enable it all additional columns will be check marked. If only additional columns are required to be exported double click the mouse on respective lines and in appeared dialog enable/disable Include field option.
Special columns	This option controls exporting all special columns*. If you enable it special columns will be check marked. If only special columns should be exported double click respective lines and in the appeared dialog enable/disable Include field option.

* During export in DB a separate record is created for every BOM item. Its fields correspond to columns of BOM blank and to additional columns. But in this case information about item type and about the section to which they belong is being lost. To save this information you may add special **Section number**, **Item type** etc. field to the record. **Special columns** option is used right for control of these fields creation.



Fig. 6.2.37. Dialog of the BOM export parameters setting. **Sections** tab

Tab. 6.2.10. Dialog of the BOM export parameters setting. **Sections** tab

Interface item	Description
Include sections	This window displays sections existing in the current BOM. The items to be exported are check marked. To exclude the section from the exported ones (or to include earlier excluded section) left click it.
Configuration	To save created BOM configuration (format, exported and non-exported sections, items, fields, etc.), enter name of new configuration in input field and click the Save button. Saved configuration may be used in further calls of BOM export into DB format command. To delete existing configuration select its name from the list and click the Delete button.

6.2.6.9. Additional Sheets

KOMPAS-3D BOM may have additional sheets at the beginning and at the end. This feature allows, for example, to create and to store change registration list with the BOM itself. Additional pages are accounted for in automatic numbering of pages.

There are the following methods of handling additional pages including their creation, deletion, setting or editing page attributes:



- ▼ in the **Document manager**, which is called by clicking the respective button on the **Standard** bar (the dialog is described in detail in Section 3.5.6.2 on p. 1182, changing page parameters,

adding and deleting — in sections 3.5.2.1–3.5.2.3);

- ▼ in the dialog for setting current BOM parameters, which is called by the **Service — Parameters... — Current BOM — Additional sheets** (see Section 9.2.5.1.3 on p. 1756).

Switch to the **Page layout** mode for additional pages to be displayed on the screen (see Section 6.2.2.9 on p. 1492). Text from the table in additional sheets is entered in the same mode.

If you use the same set of additional sheets in the majority of BOMs it is not efficient to make appropriate settings in every document. In this case you may create all new BOM with required additional sheets.

To do this, call the **Service — Parameters... — New documents — BOM — Additional sheets**.

6.2.6.10. Sheet Numbering

Setting up the current BOM sheet numbering is done in the **Sheet Numbering** dialog called with **Service — Parameters... — Current BOM — Sheet Numbering**. Working with the dialog is described in Section 9.2.4.6 on p. 1753.

6.2.6.11. The Setting of Displaying the Mass Values

It allows to set mass values and display accuracy to use in the corresponding BOM columns, e.g. in the *Mass* column of sheer BOM or in the additional column named *Mass*.

In order to setup displaying mass for a new BOM document, call the **Service — Parameters... — New documents — BOM — Display of values**. Select mass units and set decimal places in the mass value in the appeared dialog.

In order to change the setup of displaying mass for an existing BOM document, call the **Service — Parameters... — Current BOM — Display of values**.

The setting of mass value displaying for graphics document or model BOM items is stored in the BOM description used by these items. To change the setting, call the **BOM— BOM Description Management** command, select the description in the appeared dialog and click the **Edit Description** button. Select mass units and set decimal places in the mass value in the appeared dialog for setting up of the current BOM description (Fig. 6.2.23 on p. 1509). If the document is already connected to BOM, then the setting can not be changed— it coincides with the BOM setting (see Fig.6.2.25 on p. 1511).

After changing the mass units in the corresponding cells of the BOM-document or BOM item, the mass values will be recalculated. Mass values will be also recalculated in the following cases:

- ▼ At transferring of BOM items between documents, if different mass value displaying settings are used in them
- ▼ At transferring of mass value between a BOM item and the document (drawing or model) connected to it, if different mass units are used in the BOM and the document.

The set decimal places is used only for mass value displaying. That is the mass value is constantly stored in the cell with that accuracy, with which it was entered by the user or received from the connected document (drawing or model), but it is displayed in the cell rounded to the specified decimal place.

6.2.6.12. BOM on drawing

BOM items being in the drawing may be located in the BOM table on the same drawing. The following is the required condition for this: the drawing should **not** have BOM of the style connected to it similar to that of the BOM items which should be located on the drawing.



Respectively the drawing on which the table with certain style is located is impossible to connect to the BOM of the same style. To make the connection possible it is required to cancel BOM location on the page.

BOM item styles are defined by BOM descriptions (please, see Section 6.1.3.4 on p. 1461) available on this drawing.

Usually the drawing has one description of the BOM. This description is the current one.

6.2.6.12.1. Location of the BOM of the current style on the drawing

To place the BOM on the drawing call **BOM— BOM on page— Show** command. The command is accessible if no BOM of the current style is connected to the drawing.

After calling of this command a table will appear above the main text. It contains the BOM items available on the drawing.

All parameters of this table (format and number of columns, header, location of items on pages, etc.) correspond to parameters of the table included into the main text of the BOM current style as *Table for BOM*.

This table is subject to all settings of the current BOM style (the number of reserved lines, sorting etc.). Briefly, this table with items is BOM of the current style without frames and own stamp.



Designation and name of the drawing affects the image of the BOM items containing code and name of the document. This influence is similar to the influence of BOM designation and name on items in it (please, see Section 6.2.2.10 on p. 1492).

When creating new BOM objects in a drawing, they are automatically included into a BOM table on the page.

Double-clicking the mouse on the table launches the slave mode of editing BOM objects. All changes made in it after closing the mode window are passed into a BOM on the page.

To move the table on the list use **BOM— BOM on page— Location** command (or **Manual placing** command from shortcut menu).



You can create a user style, for example, for a table of connections or a listing of elements, and then place the table of connections on the page with the wiring diagram or the listing of elements on a page with a kinematic diagram. If, when working with the scheme, the respective geometry-containing objects will be created, then later this will help searching for a wire connector image or a diagram element.

To disable BOM image on the scheme call **BOM— BOM on page— Show** command again.

6.2.6.12.2. Placing of BOM of different styles on the drawing

BOM— BOM on page— Show command allows placing only one BOM on the page— the BOM of current style.

To enable location of BOM of other styles (if the drawing contains several descriptions of BOM) call dialog of BOM description control (Fig. 6.2.38). For this use **BOM— BOM description control...** command.

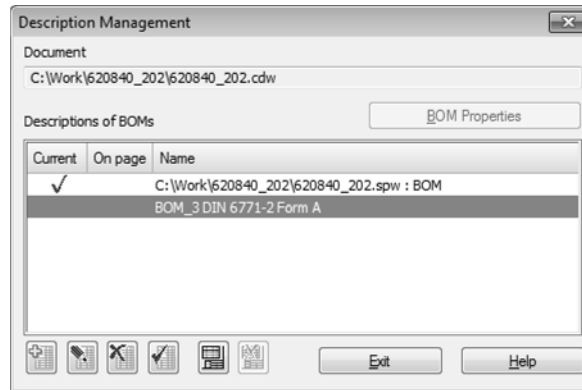


Fig. 6.2.38. BOM Description Management Dialog

In the list select the description containing required style and click the **Include image on the page** button. This button is accessible only for those descriptions which don't contain BOM file name.

If the BOM is placed on the page checkbox is displayed in **On page** column against the respective BOM description.

Close BOM description control with **Exit** button. BOM table containing BOM items of selected style will appear on the drawing page.

Key **Disable image on page** of BOM description control dialog allows canceling location of BOM on the page.

6.2.6.12.3. BOM name of the drawing

BOM name is the text above the table of the BOM located on the page. The BOM name may consist of several lines. Name text is entered by user. The user can set various names for the first and other sheets of BOM.

Name may be entered only for the BOM located on the drawing (please, see Sections 6.2.6.12.1 and 6.2.6.12.2). To define BOM name do the following.

1. Call **Name...** command from **BOM— BOM on page** menu or from shortcut menu of BOM on the page.
2. In the displayed dialog enter names for the first and the following BOM pages. If the length of entered line exceeds the BOM table width, automatic narrowing will automatically occur. To move to a new line, press the <Enter> key. If required, you may change any parameters of the text editing: font, height, leveling and indentation units, etc.

3. Close the dialog by click the **OK** button. Name (names) will appear above the table (tables).



Splitting of BOM table on pages is made in slave mode of operation with BOM using **Place on a new page** switch (please, see Section 6.2.6.5 on p. 1521).

In the future in order to edit existing BOM name call **Name...** command again. You may double click the BOM name.

BOM name is specified for the specific BOM style and is saved in the drawing.



If during setting of the system in **Text editor — Text templates** section **Save entered text in the file** option is enabled, manually entered names for BOM pages on the drawings will be automatically saved in text template file. They will be located in **BOM— BOM name on drawing page** section of this file (please, see Fig. 6.2.13 on p. 1481).

In the future to call Librarian of text templates and using of saved names double click the mouse in the text input field of the BOM name defining dialog.

BOM— BOM on page— Name... command allows working with the name of the current style BOM. If tables of different style BOM are reflected on the drawing page to enter and edit their names use **Name...** command from shortcut menu of these tables.

Distance from the BOM to the name is similar for all BOM tables reflecting on the page. It is defined in the dialog of BOM name parameters setting on the page. Default distance is 5 mm. In the BOM name parameters setting on the list you may also set default parameters of the text name. To call the dialog, use the **Service — Options... — Current Drawing — Document Parameters — BOM name on the sheet**.

6.2.6.13. Revision table

To enter information about changes into the current drawing do the following.

1. Assign to the drawing one of the variants— Drawing of str. s. *First Sheet*. GOST 2.104-2006 or Drawing of str. *First Sheet*. GOST 2.104-2006 (TCZ— table of changes with zones). You may also select similar templates for further pages of the drawing.



2. Call the command **BOM— Add object of changes table**.
3. In the appeared text part input window fill in cells and click the **OK** button.

The line filled in by you will appear in the main table of the main drawing text.



To edit lines of changes table call command **BOM— Edit objects of changes table**.

The system will switch to the slave mode of item text part editing.

Methods of work in it are similar to those used during editing of BOM items in the drawing.

Table of changes is filled bottom-upwards. Lines in it are sorted in number ascending order.

Geometry from the drawing may be connected to the item of the table of changes.

6.2.6.14. Items of different BOM in one drawing

When creating the first BOM item in the drawing this item will be created according to the style defined in the system of new BOM. The drawing will automatically receive description of BOM (please, see Section 6.1.3.4 on p. 1461). The following items will be created with the same style.

In the BOM description control dialog you may create more than one description of the BOM (please, see Section 6.2.4.1.1 on p. 1508). One document may only have BOM of different styles connected.

Description check marked in the **Current** column of the description control dialog is considered the current one. The style included in it is considered the current BOM style.

BOM items in the drawing are created according to the style included into the current description and are transferred to the document-BOM included into the current description (if connected BOM exists).

If you call BOM item creation command from the drawing the window of new item text part input will be displayed containing columns and header corresponding to the blank of the current BOM style.

In the slave mode of editing objects you can only see the objects created according to the current BOM style. After you made the other description the current one (for this select it in the dialog of description control and click the **Make the current one** button), items corresponding to the current style will be displayed; in slave mode the BOM items of current layer will be displayed (in this case BOM items of other styles will not be deleted but will not be displayed).

Synchronization commands work with a BOM that belongs to the current description.

By default the current style BOM is located in the drawing. If it is required to show BOM of other styles on the drawing, call description control dialog and enable showing of BOM for required descriptions in it (please, see Section 6.2.6.12.2 on p. 1532).

Creation of items in the drawings for BOM of different style and their transfer to different documents principally differs from the BOM style changing. During style changing the same items of one BOM may be displayed in the other blank form, go to section with different headers, have the other contents of columns (please see detailed description of BOM style change in Section 6.3.3 on p. 1560). Meanwhile, when working with different descriptions, autonomous groups of objects are created, each of which is passed into a separate BOM document.

Generally speaking, working with several descriptions of BOM in one drawing is not a standard methods of BOM creation. You shall proceed in that way only when creating of several BOM based on one assembly drawing has well-defined grounds.



Create two BOM descriptions in assembly drawing — the BOM itself and list of purchased parts. After you enter items in it these items may be transferred to two different documents — to BOM and list of purchased parts.

6.2.6.15. Items-«doubles»

Items-«doubles» — are items of one section containing similar data in the column where sorting takes place. They may appear as the result of copying of items, insertion into drawing or BOM of similar parts from the Library of Standard Items, input of similar text part, etc.

When using the automatic sorting, those objects are assigned the same position reference numbers, and all of them, except one (the first created) are hidden from the display of objects in the table. Therefore, in the usual mode such objects appear as one BOM object.



You may at any moment enable or disable the mode of showing of all BOM items (please, see Section 6.2.6.7 on p. 1522). In this mode all objects are shown in the BOM table, including those whose display was turned off by any of the means (automatically or manually).

If printing the BOM is initiated from that mode, then all objects will be printed.



If the image of a BOM object (for example, part) is located on the different sheets of an assembly drawing, create several objects with the same text portion and connect to each of them the respective graphical objects from one of the assembly sheets (including a position number leader).

In this case, you will be able to easily find the part geometry on any sheet of an assembly (to do this you need to enable the display of all objects and the display of object geometry).

Meanwhile, when recalculating position reference numbers (for example, due to new object creation), the numbers on the part leaders will be changing on all sheets.



If there are several groups of identical standard items in an assembly drawing, create a BOM object containing a position leader for each of them. For example, two different caps are fastened with the same screws.

Create an object with a position reference number leader for the screws of one of the caps and an object with a position reference number leader for the screws of the other. Those will automatically receive the same number (and the identical position reference numbers will be on the leaders). In the mode of viewing all objects and displaying geometry of BOM objects, you will be able to easily find each group of screws.

If you change, for example, the diameter of screws fastening one of the caps then the BOM objects will change and automatically receive different reference numbers (according to their new sorting order), will be visible in the BOM in the regular mode, and their changed reference numbers will appear on their leader lines. You will need only to change numbers in *Quantity* column.

6.2.6.16. Using marks/position designations

Marks/position designations — alphanumeric characters of items represented on drawings. Marks/position designations may be considered analogs of specified positions the distinctive feature of which is availability of the **mark**— text between position number.

Marks/position designations in graphical documents are created with the following commands:

- ▼ **Mark/reference designation with a leader,**
- ▼ **Mark/reference designation without a leader,**
- ▼ **Mark/position designation on the line.**

These commands are described in detail in Sections 3.3.13.3-3.3.13.5.

BOM items may have the **mark** — text before the position number. Mark is placed in *Position* column together with the number. Mark is defined for BOM section and is automatically assigned to every base item of this section.

To include using of the mark in BOM section, do the following:

1. Call **Format— BOM setting** command.
2. In the displayed BOM setting dialog go to **Sections** Tab.
3. Select required section in the list and click the **Edit section style** button.
4. In the style setting dialog appeared in the section screen (please, see Fig.6.3.3 on p. 1552) enable **Mark** option and enter its text.

When **Mark** option is enabled **Independent position numbering** option becomes enabled. It allows beginning again numbering of the section item positions— from the number specified as the starting position.

5. Close dialog of the section style setting, and then— dialog of BOM setting with **OK** button.
All items of the selected section will become of the defined mark. It also will be added to new items of this section.



Blocks of additional BOM sections may have own mark and independent numbering of sections. Setting of position mark and numbering for the block is made in dialog of sections block settings (please, see Fig.6.3.8 on p. 1557).

Similar to ref.no. leader the mark/position designation may be connected to BOM item having it included into this item content. Consequently marks/position designations will be assigned with position numbers and marks from the BOM. This transfer will have place during synchronization of the BOM with the drawing.



Digits in position mark/contents may work as not only position numbers, but as element number in the project. That is why transfer of position numbers from BOM into mark/position designations may be undesirable.

To cancel assignment of new numbers to BOM items disable position calculation in BOM (please, see Fig.6.3.1 on p. 1547). Consequently items will be sorted out according to the rules defined in the style, but position numbers will remain the same. Changing of numbers made manually would be transferred both from the drawing to the BOM and from the BOM to the drawing.

During mark/position designation connection to BOM items designation mark is replaced with BOM item mark, and manual mark replacement on the drawing becomes impossible.



As distinct from the position designations, the same mark/position designations can't be connected to several BOM items.

6.2.6.17. Position Designations in Assembly Model

Similar to a drawing, you may also create position leaders within an assembly.

The well-defined assembly structure makes it possible to automatically integrate positions into the BOM items connected to parts and subassemblies. Use one of the following methods to automatically include positions into BOM items:

Method 1 (for components)

1. Create assembly components with BOM items and insert them into the assembly.
2. Specify position leaders Each position leader will be automatically included into the BOM item connected to the component it is associated with. This shall enable display of position numbers imported from BOM items on the position leaders.

Method 2 (for components and bodies)

1. Insert components into the assembly and build the necessary bodies.
2. Specify position leaders for components and bodies.
3. Create BOM items by selecting components (or bodies) and calling the **BOM – Add Items – Internal** command (for more details See description of method No.4 in Section 6.2.3.4.2 on p. 1498). Each BOM item will automatically include a position leader pointing at the highlighted component (or body).

Irrespective of the method used for automatic inclusion of positions into the BOM items the following rules apply:

- ▼ If the position leaders point at different components of the same subassembly, all these position leaders are included into the same BOM item (corresponding to the subassembly) and are assigned the same position number.
Similarly, if the position leaders point at different parts of the same body, all of them are included into the same BOM item (corresponding to the body) and are assigned the same position number.
- ▼ If different branches of the position leader point at different assembly components, the position leader is included into several BOM items (corresponding to the components), with their position numbers appearing on the leader ledges.
- ▼ If an assembly contains multiple insertions of the same component, it is populated with the so-called “twins” of BOM items (see Section 6.2.6.15 on p. 1534).
 - ▼ If the insertions are specified by different position leaders, each of the leaders is included into one of the twin items. Since the twins have identical position numbers, the insertions are assigned the same position number.
 - ▼ If the insertions are specified by different branches of the same position leader, the position leader is included into each twin item.

To ensure that the position leaders are included into internal BOM items of the assembly, view the geometry of these items (see Section 6.2.3.5.2 on p. 1501).

If the BOM items connected to components have been created after insertion of position leaders using the **Create BOM Items...** command or manually in the component files, no automatic inclusion of position leaders into BOM items is performed. In this case, you can populate the items with positions manually (see Section 6.2.3.5.1 on p. 1501).



The position leaders which are part of BOM items are marked in the Design Tree with “SP” letters in parentheses. The letters precede the name of the position leader, for example: “(SP) Ref.No. Leader:5”.

Position leaders are transferred into the associative assembly drawing similar to other designations. If the position leader has been included into the BOM item in the assembly, it is also included into the same BOM item in the drawing.

6.2.6.18. Merging BOM description variants in a document

One document can include several BOM descriptions that contain similar BOM styles. Similar styles are styles that are taken from libraries with the same file name (for example, *Graphic.lyt*) and had the same number in such libraries. In this event, when all parameters of similar styles match they are treated as **duplicates**; when they are different (e.g., have a different number of additional columns), they are treated as **variants**.

Several descriptions with similar BOM styles can appear, for example, in an assembly when inserting parts created in KOMPAS-3D versions that contained different editions of the same BOM style. As a result, as many groups of BOM items are created in the assembly as descriptions. These groups can only be viewed one by one. By merging BOM description variants, you can bring all groups into one.

Description variants are merged in the BOM description management dialog called from **BOM – BOM description control**. If the document includes descriptions with duplicates or variants of one BOM style, the **Merge variants** button appears in this dialog and can be used to bring all these descriptions into one. In addition to **Merge variants**, the **Variant** column appears in the dialog and indicates whether each style is a duplicate or variant in relation to the sample (Fig. 6.2.39). A sample is a style included in the first description in the group of descriptions with similar BOM styles. The sample style has a dash in the **Variant** column.

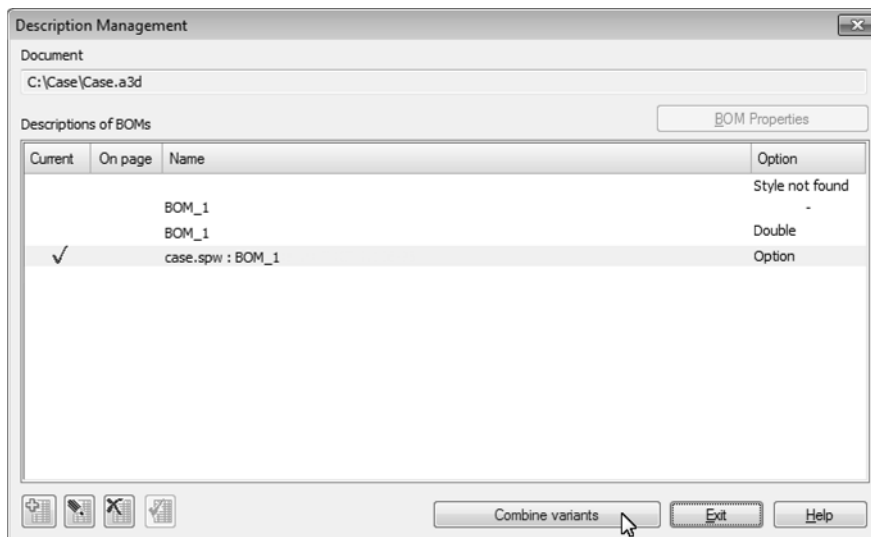


Fig. 6.2.39. BOM Description Management Dialog

When **Merge variants** is clicked:

1. If there are descriptions with an unfound BOM style (i.e. the style whose number is absent from the library that this style was taken from or a style for which no library has been found), the

user is prompted to replace the style in each of these descriptions (Fig. 6.2.40). After **Yes** is clicked in the dialog, the dialog for indicating the style library and selecting a style from it appears on the screen.

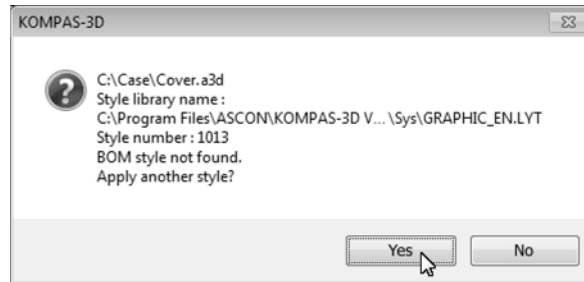


Fig. 6.2.40. Request for BOM style replacement

Please note that the merging is terminated when you cancel style replacement.

2. Replacement is followed by a check whether there are any duplicates or variants of the sample style among specified styles.
3. Bringing BOM styles in descriptions in line with the sample style.
 - 3.1. In descriptions containing the duplicate or variant of the style, information about his style is replaced with information about the sample style.
 - 3.2. BOM items that used the duplicate or variant of the style are converted into BOM items with the sample style. Here, the same changes take place in the item as in the B at style change. See Section 6.3.3 on p. 1560.
 - 3.3. BOM items that used descriptions with duplicates or variants of description are moved to the group of items that use the description with the sample style.
 - 3.4. Descriptions no longer used in BOM items are removed from the document.



If a pair of identical BOM items where one “comes” from a component file and one was automatically created on the basis of this component's properties (see point 2 in Section 6.2.3.4.2 on p. 1498) are found in an assembly when BOM descriptions are merged, the latter is deleted.

Changes described in point 3 are also performed in source files of BOM items if items have been transferred to the current document from other files (fragments or components). If any of the sources are currently unavailable, a message appears on the screen when processing the first one of them that description merging failed (Fig. 6.2.41). By clicking the respective button in the message, you can save the list of unprocessed documents to a file or cancel saving. In any case, description merging is terminated, i.e. the remaining files are not processed. Eliminate problems in BOM item source documents and then try to merge BOM descriptions again.

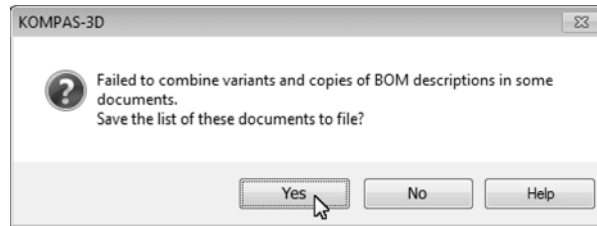


Fig. 6.2.41. Message saying that BOM descriptions cannot be merged

After descriptions are successfully merged the respective message appears on the screen. In the description management dialog, each group of descriptions that contained duplicates or variants of one BOM style is replaced with one description with a sample style.

Save the document after BOM descriptions are merged.

6.2.7. Storing data in documents on used BOM style

The style of the document-container, as well as the style included in the document BOM description (see Section 6.1.3.4 on p. 1461), are embedded into the document. As a result, you can work with BOM items even if there is no *.lyt library containing the BOM's layout. Editing the BOM style saved in a document is not possible.

In addition, a link to an external library file *.lyt containing the selected layout is stored in the document. The link includes path to the library and its number of style. Therefore, when design is changed in the library, the document's design can be updated.

Note that when you update the BOM layout, designs that are used in it (together with the title blocks included) are also updated automatically for the first page and for the following pages. Layout update separately from the style is not possible.

The style update depends on the layout update settings, see Section 10.2.1.2.1 on p. 1961. A forced BOM design update is possible at any moment when working with a document, see Section 6.2.7.1 on p. 1541.



If there are additional pages in the BOM-document (see Section 6.2.6.9 on p. 1529), designs of these pages will also be embedded into the document and updated using the same setting.



All links available in the current document, including the links to the *.lyt libraries, can be viewed on the page **External Links** of the document summary dialog. Use the **File – Document summary...** command to activate this dialog.



Starting with the 13th KOMPAS-3D version BOM style is embedded to a document. If a document was created in an earlier version, you will not be able to work with BOM items within the document if there is no *.lyt library containing the BOM layout used (the BOM-document will just not open).

6.2.7.1. Style re-reading

Layout Re-reading— forced document layout update.

Style re-reading is available:

- ▼ **In a BOM document**, You do this using **Service— Re-read Style** command. It is available if there are differences between style stored in a document and respective style in the *.lyt library. If there are no differences and also if there is no library or no style in it (i.e. it is impossible to compare styles), the command is unavailable.
- ▼ **In the current description of the BOM of a graphical document or a model**. To do this, call the **Service — Re-read style** command when working with the BOM in slave mode (see Section 6.1.3.3 on p. 1461).
- ▼ **In the current description of the drawing's BOM and in all BOMs on the drawing**. To do this, call the **BOM— Re-read Styles** command.
- ▼ **In a single BOM on the drawing**. To do this, call the **Re-read Style** from shortcut menu of BOM on the drawing.



If a BOM contains additional pages, their layout will be updated when the BOM style is re-read, i.e. when the **Service— Re-read style** command is called.

You can also manually update layouts of additional pages in the BOM's beginning and/or end. To do this, call the **Service — Parameters... — Current BOM — Additional sheets**. In the right-hand part of the dialog select a layout template and click the **Change** and in next dialog— **Re-read** button. Close the setup dialog by clicking the **OK** button.

6.3. Custom BOM Settings

6.3.1. Style of the BOM

The exercises in the previous chapter have helped to learn some parameters and settings of a BOM.

You changed the number of reserved lines in the sections, enabled and disabled the automatic sorting of items, enabled and disabled the position numbers placement.

You have also created BOM of different styles, i.e. *simple* and *group*. You may have noticed that the BOM of different styles may vary in appearance and be different in their content.

The style of the BOM is a set of BOM-specific parameters and settings affecting its content and representation.

6.3.1.1. Style Components

The BOM parameters and settings can be subdivided into six groups.

1. BOM template:

- ▼ Sheet size.
- ▼ Borders.
- ▼ Title block (form) of the first and subsequent sheets of the BOM generated in a special way.
the first and other sheets of BOM.

2. Common settings:

- ▼ Presence or absence of a connection between an assembly drawing and the BOM and the type of this connection.
- ▼ Automatic position numbers calculation (enabled or disabled).
- ▼ Zone calculation (enabled or disabled).
- ▼ Delete geometry when deleting related item from BOM (enabled or disabled).
- ▼ Delete BOM item when deleting related geometry (enabled or disabled).
- ▼ Copy a BOM item when copying related geometry (enabled or disabled)
- ▼ First BOM item position number.
- ▼ Direction of building the BOM (top-down or bottom-up).
- ▼ Number of variants of the specified product.
- ▼ Path to the text template file where the entered texts will be saved.

3. BOM form columns.

The number of columns corresponds to the number of columns in the BOM table (which is a part of the title block). **For each column** of a form you can specify:

- ▼ Name.

- ▼ Type.
- ▼ The number of the column of this type.
- ▼ Column data type.
- ▼ Presence or absence of a link to the title block columns of the drawings connected to the item, tangent to object.
- ▼ Presence or absence of single-line text alignment to the bottom line of the BOM item BOM in the given column.
- ▼ Possibility of entering text into the column when creating BOM items in a model.

If a column contains integer or real number data type, the feature of summing up the data in the given column (with or without multiplying the data by a number in the *Quantity* column) is enabled or disabled.

The display of section titles is enabled in one of the columns. It will contain all section titles of the BOM.

Each column settings are set by default for the entire BOM.

4. Additional columns.

The additional column list is generated by the user. **For each** you can specify:

- ▼ Name.
- ▼ Type.
- ▼ The number of the column of this type.
- ▼ Column data type.
- ▼ Presence or absence of a link to the title block columns of the drawings connected to the item.
- ▼ Possibility of entering text into the column when creating BOM items in a model.

When customizing an additional column that contains numbers, the feature of summing up the additional parameters in the given column (when multiplying by a number in the *Quantity* column or without doing it) can be enabled.

Each additional column settings are set by default for the entire BOM.

5. Sections. This is the most extensive block of settings.

The list of sections is generated by the user.

The breaking of the BOM into sections and displaying the section titles can be enabled or disabled.

For each section you can specify:

- ▼ Name.
- ▼ Number.
- ▼ The number of its reserved lines.
- ▼ Sorting type (rules).
- ▼ Whether the position numbers setting in the given section is enabled or disabled.
- ▼ Possibility to attach geometry to section items.
- ▼ Section location — on a new sheet or arbitrary.

- ▼ Presence or absence of a section mark. If there is mark present— its text is entered and the section item independent numeration is enabled or disabled.
- ▼ Sorting the items in a section.
- ▼ Division into subsections, list of subsections.
- ▼ The use of nested sections blocks is enabled or disabled.
- ▼ Method of columns filling in the section (manual only or with sending data from connected documents enabled).

The columns of a form and the additional columns in a given section can be customized so that their filling rules differ from the default ones (defined at setting-up as proper and additional columns):

- ▼ Enable or disable the filling of each particular section column.
- ▼ Change the data type in a column.
- ▼ Enable or disable the column cells link to a particular column of the title block of the document, connected to the item.

If the use of nested sections blocks is enabled, one can specify whether the block titles display is necessary in the BOM. The list of blocks is generated by the user. For each block you can specify:

- ▼ Name.
- ▼ Number.
- ▼ Whether it is used by default.
- ▼ Block displaying method — from the new sheet or arbitrary.
- ▼ Sections list.

6. Blocks of variants

This group of settings defines the rules of generating and displaying the variant numbers (for both the specified product and the BOM items).

It allows to:

- ▼ Enable or disable the display of the variant blocks titles.
- ▼ Select a method of block displaying — on a new sheet or arbitrary.
- ▼ Enable or disable automatic creation of notes about the absence of variable data in versions (only for variant A of group BOMs).
- ▼ Enable or disable the addition of zeroes and/or dashes before the product variant number.
- ▼ Select a BOM display method (in blocks or each item separately) if the number of product variants exceeds the number of columns for entering quantity per variant.
- ▼ Specify the method of displaying the item-variants designation (entire or variant numbers only).
- ▼ Specify whether the item-variants position numbers should ascend.

7. Blocks of additional sections.

Enable or disable the blocks of additional sections in a BOM.

If blocks of additional sections are enabled, one can specify whether the block titles display is necessary in the BOM.

The list of blocks is generated by the user. For each block you can specify:

- ▼ Name.
- ▼ Number.
- ▼ Sections list.
- ▼ Whether it is used by default.
- ▼ Block location — on a new sheet or arbitrary.
- ▼ Presence or absence of a block mark. If a mark is present— its text is entered and the independent numeration of the items is enabled or disabled.

Some of these parameters and settings can be modified for each particular existing BOM. This is detailed in Section 6.3.2 on p. 1546.

Other settings and parameters (for example, locate the items of the current BOM on the other form) can only be modified by switching the BOM style to the one with required parameters and settings. This is detailed in Section 6.3.3 on p. 1560.

If a BOM style with required parameters and settings (for example, user BOM form) does not exist yet, it can be created by the user. This is detailed in Section 6.3.4 on p. 1561.

The methods of customizing the appearance and properties of an existing BOM (modifying style and BOM setup) can be combined (for example, modifying BOM style with further setup of parameters to differ from the default parameters of the given style).

6.3.1.2. Using BOM mechanisms for creating other documents

Just as the document drawing (file **.cdw*) can contain a diagram, a poster apart from the drawing proper, a document BOM (file **.spw*) can contain a list, a register, etc.

The variety of parameters and settings, especially the opportunity to employ user forms, allows one to create documents other than BOM according to GOST. The mechanisms of the BOM design system are suitable for processing various registers, indexes, catalogs and lists.

Generally, a document BOM is a table filled with heterogeneous information per line and having a dedicated location (form column, additional column) for each data type. The resulting lines can be numbered, sorted, linked to documents and graphical items, etc.

Combining various settings, one can create BOM styles for BOM sheets, reference documents sheets, purchased product sheets, connection tables, revision sheets and other documents.

6.3.2. Customizing an existing BOM

A BOM appearance and filling rules depend on the style of that BOM.

The BOM style consists of a large number of similar parameters and settings (see Section 6.3.1 on p. 1543).

Parameters and setting that do not require changes of BOM form or modifications in the structure and content of BOM item can be modified individually for each existing BOM.

Those are the general settings and some of the section settings — number of the reserved lines, setting the position numbers in a section, attaching geometry, sorting (enabled or disabled).

Besides, nested sections blocks can be enabled or disabled for each section, and for the whole BOM— blocks of additional sections. If the blocks are in use, they can be customized (enable or disable the availability of certain blocks and their sections, etc.).

The modification of these parameters is called **customizing of the BOM**.



As you know, some settings (for example, the number of reserved lines in a section or the need to sort its items) are easily modified using fields and buttons on the **Current Status** bar.



To modify other parameters of the current BOM use the **Format— BOM Properties** command.

If this command is called from the slave mode of the BOM items editing, then the modified settings influence both the BOM connected to the current drawing and the slave mode itself. The BOM Properties command is not available in the slave mode if the document BOM is not connected to the current model or drawing.

After calling the command, BOM setup options dialog appears on the screen (Fig. 6.3.1).

Its upper portion displays the name of the current BOM style. The dialog contains three pages— **Settings** (see Section 6.3.2.1), **Sections** (see Section 6.3.2.2), **Variant Blocks** (see Section 6.3.2.3) and **Blocks of Additional Sections** (see Section 6.3.2.4).

6.3.2.1. General BOM settings

The control elements of the page **Settings** (Fig. 6.3.1) modify the general BOM settings (make them different from the settings provided by the BOM style).

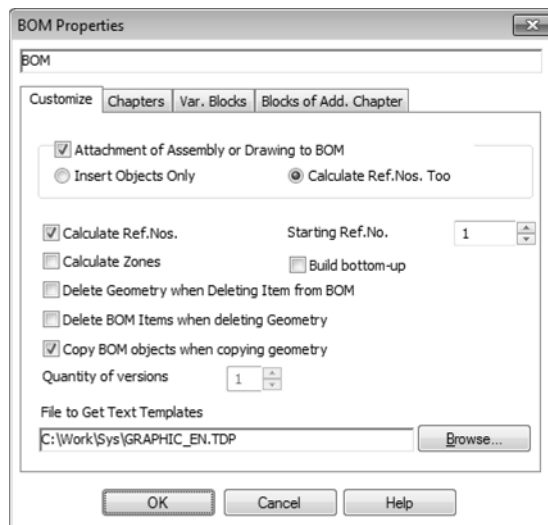


Fig. 6.3.1. General BOM settings

Tab. 6.3.1. Control elements of the page **Settings**

Element	Description
Link of the Assembly or Drawing with the BOM	<p>Group of options that enables or disables the connection between the attached documents, i.e. BOM and model (assembly drawing) and sets up the type of this connection.</p> <p>If an option Attachment of Assembly or Drawing to a BOM is disabled (for further details see Section 6.3.2.1.1), the sending of items from attached drawings into a BOM is impossible*.</p> <p>With this option enabled, the following connection options become available — Insert BOM Items Only and Calculate Position Numbers.</p>
Insert Items Only BOM Table	<p>If this connection option has been selected, the items are sent from the drawing into the BOM and are sorted. New position numbers are not assigned to them. It may result in items arranged not in ascending order of their position numbers. The position numbers are not sent to the leaders in the drawing.</p> <p>It is recommended to use a connection without position numbers calculation when the position numbers in the drawing are letters.</p>
Link with calculation of positions	<p>With this connection option selected, the items are sent from the drawing into the BOM and are sorted with the new position numbers assigned to them. After that, the renewed reference numbers are sent to leaders in the drawing.</p>
Calculate positions	<p>This option enables or disables the calculation of the position numbers (i.e. reassigning position numbers to the items after the ordering has been disrupted due to the automatic sorting or deleting of items). With this option enabled, there is no automatic position number calculation (after each sorting operation). To perform calculation, call the command Set Position Numbers (see. Section 6.2.2.8 on p. 1491).</p>
Starting position	<p>This is the field for assigning the position number to the first BOM item in the first section, for which the Set Reference Numbers is enabled (see Section 6.3.2.2.1 on p. 1552).</p>
Calculate zones	<p>This option enables or disables the sending of the names of the assembly drawing zones for item position numbers leaders into the BOM. If enabled, name sending takes place after calling the Set Position Numbers command, as well as during synchronization of drawing and BOM (see Section 6.2.4.2.1 on p. 1513).</p>
Build Bottom Up	<p>This option manages the sequencing of the sections and the items in them. With this option enabled, the sections are arranged in reverse order to that specified in the BOM style and the item is also sorted in “reverse” order. Such order is recommended by some Company BOM for filling a BOM that is arranged on a drawing.</p>

Tab. 6.3.1. Control elements of the page **Settings**

Element	Description
Delete Geometry when Deleting a BOM Item	This option manages the deletion of graphical items that are part of BOM items to be deleted. With this option enabled, the deletion of a BOM item results in deleting the related graphical items from a drawing (see Section 6.2.6.1.1 on p. 1518).
Delete BOM Items when Deleting the Geometry	This option manages the deletion of BOM items when deleting related graphical items. With this option enabled, the deletion of the BOM item geometry from the drawing results in deleting that item from the BOM (see Section 6.2.6.1.2 on p. 1518).
Copy BOM Items when Copying the Geometry	This option manages the copying of BOM items when copying related graphical items. With this option enabled, making a copy of the BOM item geometry in the drawing results in making a copy of that item in the BOM (see Section 6.2.6.1.3 on p. 1519).
Quantity of Variants	This field is available when customizing group BOM only. The number of variants of the specified product may be entered therein. The entered value will influence of the availability of cells <i>Quantity per variant</i> and the number of cells.
Document	This reference field appears in the dialog when customizing group BOM only.
File name Text Templates	This field represents the name (with path) of the text template file (*.tdp), which contains texts to be used when filling and editing text of current BOM items (see Section 6.2.2.4.8 on p. 1480). To select another file, click the Browse... button. The text template file specified in BOM setting options dialog may differ from the predefined text file specified at the text editor setup.

- * It is important to understand that disabling this option doesn't break the connection between a BOM and an assembly drawing (i.e. it doesn't disconnect them), it just blocks the sending of items from the drawing into the BOM while editing the drawing. In this case, all the service capacities determined by the link between the assembly drawing and the BOM remain available. For example, the BOM items are available for viewing in a geometry drawing.



If options **Calculate Position Numbers** and **Calculate Zones** *are disabled* when you setup a BOM, the command **Set Position Numbers** is unavailable.

6.3.2.1.1. Disabling the attachment of an assembly to a BOM

When does disabling the option **Attachment of Assembly or Drawing to a BOM** make sense?

Sometimes the assembly drawing comprises several sheets. Here several people (each of them in his/her own file) can work simultaneously on the files, containing these sheets and the attached BOM.

With the option **Attachment of Assembly or Drawing to BOM** enabled, all changes made in the BOM items on the drawing will be sent into the attached BOM each time when saving or closing the drawing. The constant flow of changes that the BOM receives from several sheets may hamper the actions of the user working with the BOM. The user must constantly confirm the saving of the received changes, get oriented in the continuously changing order of and the items, etc.

To get rid of these unproductive troubles, it is possible to disable the option **Attachment of Assembly or Drawing to BOM** for the time of team work (it is possible for both the user working with the BOM itself and the user working with any connected drawing). After creating all the items in the assembly drawing, one can enable the connection option and send (synchronize) data from every sheet to the BOM.

6.3.2.2. Customizing BOM sections

Some settings of every section of the current BOM can be modified to differ from the default (style-specific) on the page **Sections** (Fig.6.3.2)








Fig. 6.3.2. Customizing the BOM sections

Tab. 6.3.2. Control elements of page **Sections**

Element	Description
Show Titles Sections	<p>This option permits to enable and disable the display of the sections and blank line titles in a BOM table.</p> <p>With this option enabled, the items are shown in the BOM table as a continuous list. At that, this doesn't break up their sorting order (their relation to the different sections is also taken into consideration). The status of this option doesn't affect the reserved lines representation.</p>
Header...	<p>This button allows to call the commands for character formatting (font, color, size) for the first and the subsequent section title lines, select the section titles aligning method (left, right or center), enable or disable the displaying of blank lines that frame the section titles.</p> <p>The settings are valid for the titles of all sections.</p>
Use the Font from Layout	<p>With this option enabled, the font of the BOM text portion will be identical to the default font of the BOM table. Disable the option to have another font.</p>
Font of the item	<p>This button allows to set up the font for the BOM item text portion with parameters different from the ones in the template.</p>
Sections	<p>This window contains the list of sections of the current BOM. Besides, it shows the sections with no items which are therefore invisible in the BOM.</p> <p>On the opposite side of every section there is a combination of icons symbolizing its settings (see Table. 6.3.3).</p> <p>To modify the settings of a certain section, select its title and click the Edit Section Style button. The Set Options dialog for the selected section will appear on the screen (see sections 6.3.2.2.1–6.3.2.2.3).</p>

Tab. 6.3.3. Icons symbolizing the section settings.

	Description
	Subsections enabled
	Data from documents attached to BOM items, are sent to the BOM
	The item sorting within section is enabled
	Geometry can be connected to section items
	The position numbers within a section are enabled

6.3.2.2.1. General section setup

In the section setup dialog (Fig. 6.3.3), you can change its attributes and completion rules (see Table 6.3.4).

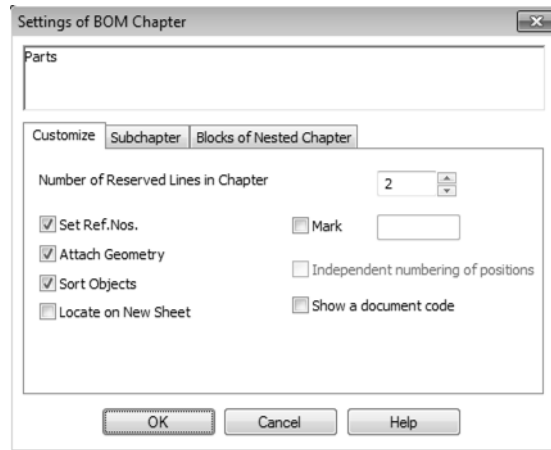


Fig. 6.3.3. Customizing a section



There is no need to call the setup options dialogs of the BOM and its section for making quick modifications of current section settings. One can use the options on the **Current status** bar (see Section 6.2.1.2 on p. 1469). They duplicate most of the options from the section setup options dialog.

Tab. 6.3.4. BOM section settings

Element	Description
Quantity reserved lines in Section	In this field one can change the number of reserved lines in the customized section.
Set positions	This option enables or disables the placement of the position numbers in a section. The unavailability of this option means that the <i>Position</i> column filling for the given section is disabled in the BOM style *.
Attach Geometry	This option enables or disables the attachment of the geometry to the section items.
Sort objects	This option enables or disables the automatic sorting of the section items.
Locate on the new sheet	This option enables or disables the placement of the section on a new sheet.

Tab. 6.3.4. BOM section settings

Element	Description
Mark	This option allows entering the text which will be automatically pasted before the position numbers of the main items of the customized section. For example, the BOM style for use in civil engineering may contain a <i>Lintels</i> section with a <i>LN mark</i> ** (see also Section 6.2.6.16 on p. 1535).
Independent Position Numbering	This option allows to enable the section's own— independent from the preceding sections—numeration. The first item of the section with independent numbering receives a starting position number (see Fig.6.3.1 on p. 1547). This option is available when the option Mark is enabled.

* One should understand that the disabling of the setting of the position numbers differs from disabling the display of the position numbers for items (see Section 6.2.2.6.1 on p. 1485). The enabled setting of the position numbers in the section with display of the position numbers disabled for some of the items will ensure the assignment of position numbers for these items at the automatic position numbers setting; though invisible in the table (because the display of the position numbers has been disabled), these numbers will be regarded at the numbering of the next items.

In case the setting of the position numbers in the section is disabled, then the *Position* column will be cleared for every item in the section and the position numbers of the next section will ignore the items and the reserved lines of the current section regardless of the enabling or disabling of the display of the position numbers.

** The text of the mark is located in the *Position* column. Therefore, even if the mark is set, disabling the *Position* column filling in the style of the BOM will result in displaying the items of the given section without a mark.

The status of the **Set Position Numbers** option does not affect the displaying of the mark.

6.3.2.2.2. Subsections setting

One can disable subsections on the page **Subsections** of the section set options dialog (Fig. 6.3.4).

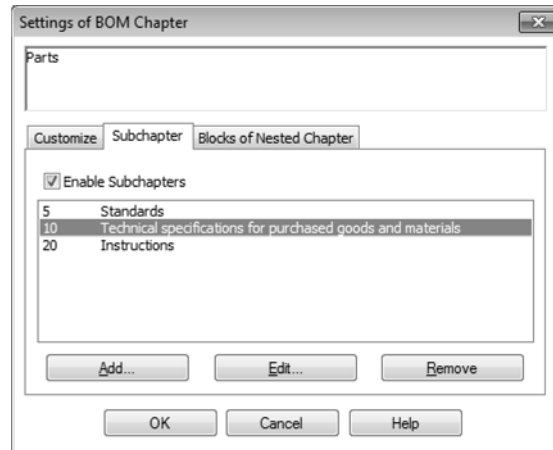


Fig. 6.3.4. Section subsections customizing

With the subsections enabled, a list of subsections can be generated. To create, edit and delete subsections, use the buttons **Add...**, **Edit...** and **Delete** respectively.

With the subsections enabled, the subsections list can be left blank. In this case, the subsections in the sections can be created “on the run”, while working with a BOM. For that, the number of subsection, to which the item belongs, should be entered in the BOM subsection property bar.

On clicking the **Add...** or **Edit...** button the subsection description dialog appears (Fig.6.3.5). Specify the subsection name and number there. The subsections are arranged in the section in ascending order of the numbers.

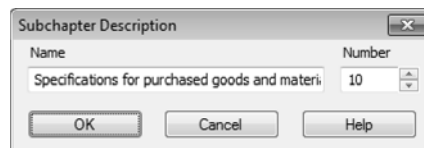


Fig. 6.3.5. Subsection description



It is recommended to issue section numbers with a reserve (for example, 5, 10, 15, 20...) rather than issuing consecutive numbers (1, 2, 3, 4).

In this case you could add a section in the middle of the BOM, if necessary, without affecting the numbering of the existing sections.

6.3.2.2.3. Customization of nested sections blocks

On the page **Blocks of Nested Sections** in the section set options dialog (Fig. 6.3.6), one can customize the work with blocks of nested sections.

The control elements for customizing blocks of nested sections are similar to the ones used for customizing blocks of additional sections (see Section 6.3.2.4 on p. 1557).

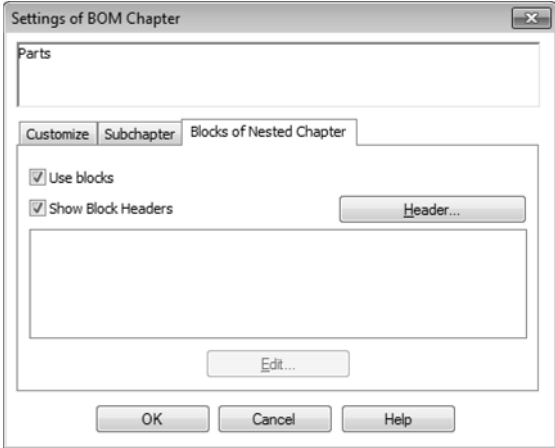


Fig. 6.3.6. Blocks of nested sections customizing

6.3.2.3. Variant blocks customization

The settings, the referring to input and display of data on specified product and the item-variants can be customized to differ from the default (style-specific) on the tab **Var. Blocks** (Fig. 6.3.7) (see Table 6.3.5).

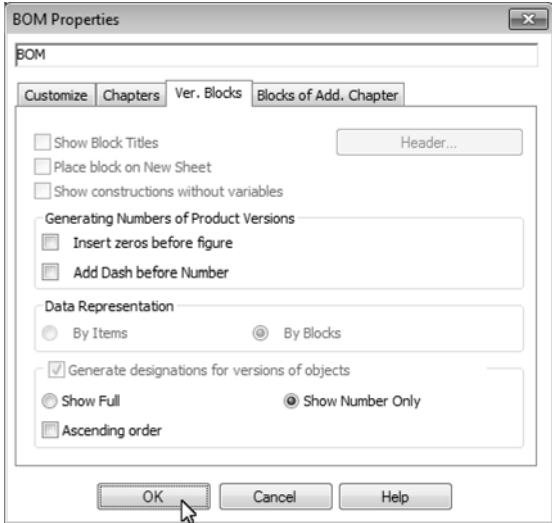


Fig. 6.3.7. Version block setup

Tab. 6.3.5. Control elements of the page **Var. blocks**

Element	Description
Show Titles Blocks	This option enables or disables the display of the variant blocks titles and the blank lines around them in the BOM table.
Header...	This button allows to call the commands for formatting characters (font, color, size, etc.) for the first and subsequent section title lines, select the section titles aligning method (left, right or center), enable or disable the display of the blank lines that frame the section titles. The settings are valid for the titles of all variant blocks.
&Place block on New Sheet	This option allows to specify whether the top of every block should be located on the new sheets of the BOM.
Show versions without variable data	This option controls processing of versions without variable data in the variant A of group BOM. If the option is enabled: <ul style="list-style-type: none"> ▼ if there is no variable data for the version, the line with the <i>No available</i> words is added to its respective block. ▼ if there is no variable data for all versions, blocks are not created, and the line with the <i>Version differences for assembly drawing</i> text is added to the end of BOM. If the option is disabled, versions without variable data are ignored.
Add Leading Zeroes to the Number, Add a Dash before Number	These are the options to customize the format of the variant numbers of a specified product (the variant numbers are located in the BOM header). Customizing the variant numbers makes sense for group BOM only.
By Items, By Blocks	If the number of variants specified in the general BOM settings exceeds the number of columns in the form for entering quantity per variant, then the display of the method selection becomes available.
Generate Designations of Variants Items	The group of controls for setup of designation and version object position display in the BOM table.
Show All, Show Number Only	These are the options to select the display method for item-variants designation (see Section 6.2.2.3.3 on p. 1473).

Tab. 6.3.5. Control elements of the page **Var. blocks**

Element	Description
Positions Ascend	By default, the numbers of the items-variants are not ascending and are not shown in the table. Enable this option if the ascending numbers of the items-variants should be displayed in the BOM.

* Display of block headers should be enabled (the **Show block headers** option).

6.3.2.4. Customizing Blocks of additional sections

The page **Blocks of Additional Sections** of the Section Setup Options dialog (Fig. 6.3.8) can be used for customizing the work with additional sections.

The description of the control elements of the page is represented in Table 6.3.6.

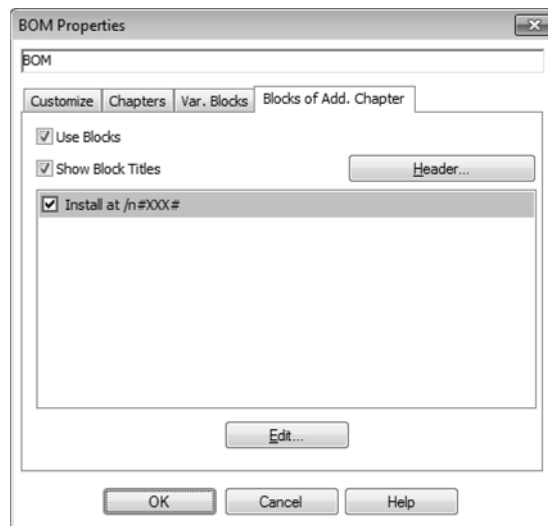


Fig. 6.3.8. Customizing blocks of additional sections

Tab. 6.3.6. Control elements of the page **Blocks of nested sections**

Element	Description
Use blocks	Option that manages the using of blocks. Enabling it makes the blocks of additional sections available when creating items in the current BOM.
Show Titles Blocks	This option enables or disables the display of the section blocks titles and the blank lines around them in the BOM table.

Tab. 6.3.6. Control elements of the page **Blocks of nested sections**

Element	Description
Header...	This button allows to call the commands for character formatting (font, color, size, etc.) for the first and subsequent section title lines, select the section titles aligning method (left, right or center), enable or disable the display of the blank lines that frame section titles. The settings are valid for the titles of all additional sections blocks.
List of Blocks	The list of additional sections is generated at the customizing of the BOM style. At customizing the current BOM, one can only enable or disable the availability of one or another block. That's what the options next to the block titles are used for. Enabling the options means availability of the block. Besides, customizing of the existing blocks is also possible. It is performed in a dialog that can be called with the Edit... button.
Edit...	This button calls the setup options dialog for the selected block (see Fig.6.3.9 and Table 6.3.7).

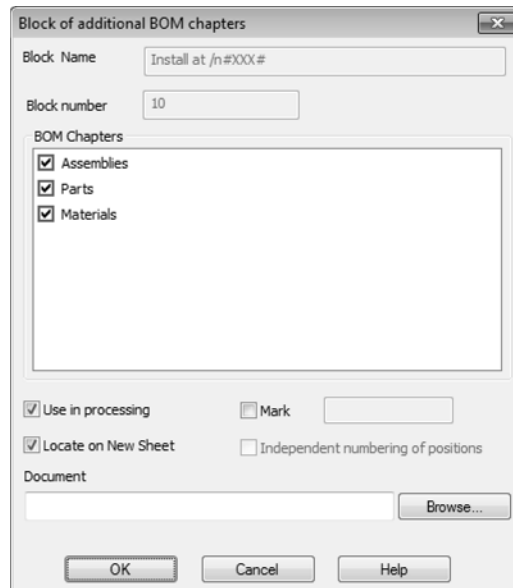


Fig. 6.3.9. Block setup options dialog

Tab. 6.3.7. Control elements of the block setup options dialog

Element	Description
Block Name and Block Number	Reference fields. Their values are defined when setting up a BOM Block Number style.
Sections BOM Table	All sections defined in the BOM style are available in the list of sections. To mark those that should be used in the block being set up, activate the option next to the section name. The section settings used in the blocks are fully identical to the settings of the respective ordinary sections.
Use in Processing	Option that controls the availability of the block being customized when filling out the BOM (the given option duplicates the corresponding option in the block list— see Fig. 6.3.8).
Locate on New Sheet	The option controlling the position of the block being set up.
Mark*	This option allows entering the text which will be automatically pasted before the position numbers of the main items of those block sections where the automatic filling out of the <i>Position</i> column is enabled. If in the BOM section, which is a part of the block, the proper mark has been set, it is ignored and the block mark is used.
Independent Numbering of Positions*	This option allows the items of the block being customized to have their own — independent from preceding items—numbering of positions. The first item of the block with independent numbering receives a starting position number (see Fig.6.3.1 on p. 1547). If the block contains a section with independent numeration, this setting will be kept, i.e. the first item of such section will have a starting position number. The Independent Numbering of Positions option is available if the Mark option is enabled.
Document	The field that contains the full name of a KOMPAS document for connection with item title. Designation from this file may be inserted into block title. To make this insertion available, the block title (specified at the customization of the BOM style) should contain a syntactic construct such as #XXX# ^{**} . This construct in the block title will be substituted for the designation obtained from the document whose path is indicated in the Document field. Therefore, specifying a document makes sense only when customizing blocks whose titles contain a construct such as #XXX#. Connection table or wiring diagram are usually chosen to be connected to the block title. To select or change the document for connection, click the Browse... button to the right from the field.

- * The option is available in the dialog only when customizing the block of additional sections.
- ** The type and the number of the symbols within “grids” make no difference.



The name of the block defined in the style may also contain one or several \n constructions. It means that the whole remaining text will be transferred to the new line. Therefore, the \n construct can be used to create multi-line block titles.

6.3.3. Modifying a style of an existing BOM

Modifying the BOM style is considered to be a more radical action than customizing the BOM. A style is usually changed to modify parameters and settings that affect the item representation in the BOM table (for example, BOM form or assortment and settings of columns).

One can change the style of the current BOM (see Section 6.2.2.2 on p. 1471).

If a BOM contained no items at the moment of the style change, then a different form of BOM will be the only visible result of the style change.

If a BOM contained items, it will be rebuilt according to the following rules.

- ▼ A newly assigned style will set up a BOM form of its own.
- ▼ The BOM item will be located in the sections and subsections that have the same number as the previous sections and subsections. For example, the items of subsection three in section five will still stay in subsection three in section five. Only the title of section five may be different than the previous one. If a number of a section is not among the section numbers of the newly set style, the items of such section will not be displayed in the BOM. It is important to understand that these items are not deleted; they are only hidden in a table as no place is assigned for them in the newly set BOM style. The setting up of a previous style will make the items visible again.
- ▼ The data will be transferred into the BOM item columns according to the column type and the number of the column of the given type. For example, column DESIGNATION(1) of every BOM item will receive data that has been stored in the DESIGNATION(1) type column of those items before the style change. In such a case, the names of those columns in the style, their titles and sequence in the BOM form are not essential. For example, the third column in a form had type POSITION(1), but in the newly set style the first column is of POSITION(1) type. The data from the third column will now be located in the first column. If a newly set style contains a column of some type (and column type number), which a previous style did not contain, then this column will be blank immediately after the transformation. It may be filled later in the regular way when editing each item. If a newly set BOM style does not contain a column of some type (and column type number), which was contained in the previous BOM style, then no data from such column will be displayed with the new style. The data will not be deleted from the item. It will be displayed when a required type column appears in the style of the BOM.
- ▼ Transferring and display of data in the additional columns is based on the same principle.
A good example for well-organized BOM transformation will be the changing of the BOM style from *Simple* to *Group* and vice versa. In these styles, the sections with identical titles have

identical numbers and respective columns — identical types. Therefore, during their transformation, all sections and the item text in them are saved.



Open any of your BOM that have been created using the system style *Simple* and change its style into *Group*. Make sure that the form has changed but the text portion of items in it remains the same.

Actually, the changing of the style of a BOM that contains items is not a typical situation. To get some meaningful results (rather than distorting an existing document), it is necessary to provide an opportunity for transformation and take into account the expected results from it when customizing the styles (sections and columns).

For example, if some section has to become invisible after transformation, then no section in the other styles should receive the number of that section. If a transformation has to interchange column positions, one should assign the same type to the different columns of the form in different styles.



One may create a BOM style for making a sheet for purchased products and assign some of its sections to the same number as the *Standardized Products* section of the BOM. After switching to the purchased products sheet BOM style, one will get a template of the sheet (the assembly units, parts, etc. will not be visible in it).



One may create a BOM style for making a BOM sheet and assign some of its sections to the same number as the *Assembly Units* section of the BOM. After changing the BOM style of the BOM sheet, one will get a template of a BOM sheet.



One may create different BOM styles with identical forms (layout) and various combinations of sections (for example, some of the section numbers match the section numbers in other styles, some section numbers are unique). Different item lists will then be displayed in the same form at transformation.

6.3.4. Creating a custom style of a BOM

6.3.4.1. General Information

The BOM styles that enable the drawing of BOM and other tabular documents according to GOST (the specific designations of the standards are shown in the style titles) are supplied as a part of KOMPAS-3D.



BOM Styles delivered in KOMPAS-3D distribution disk are stored in files *Graphic.lyt* and *Vector.lyt* from folder ...\\Sys.

To create a BOM with different rules of filling and layout (for example, according to a company standard) it is essential to provide it with a style that contains all necessary parameters and settings.

Section 6.3.1 on p. 1543 explains the parameters and settings that can be modified when generating a user style of a BOM.

It has to be noted that the new BOM style is usually generated for creating documents in case the needed form is not in the existing styles. It is pointless to create styles that differ only in parameters that can be modified for each individual BOM (for example, number of reserved lines in section).

Nevertheless, if a style that is to be created has much in common with the existing one, the latter may be used as a prototype. This is possible if the new style is created in the same library to which the prototype belongs.

Prior to creating a new style, select its prototype. Answer “Yes” when the system asks about using the selected style as a prototype.

The new style will have similar settings as the style prototype— you’ll only have to modify them.

6.3.4.2. Standard sequence of operations when creating a user style of a BOM

1. Prior to creating a new BOM style, it is necessary to deliberate on all of its settings.
 - 1.1. What form will the new BOM use? What tables are needed for entering BOM items and filling its title block?
 - 1.2. What sections and subsections will it have? What properties will every section have (sorting and numeration of items in it, etc.)?
 - 1.3. What columns will it have?
 - 1.4. What type of data will be used to fill the columns in every section? Which section columns should stay blank?
 - 1.5. What additional columns are needed in the BOM? What type of data will be used to fill them? Will summing of the values in them be necessary?
 - 1.6. Should the BOM support an associative link to drawings and models?
 - 1.7. Is calculating of reference numbers and zones required?
 - 1.8. Will the BOM be group one? If so, which method will it use to represent information on product variants?
 - 1.9. Will it be necessary to create items-variants in the BOM?
 - 1.10. Are blocks of additional sections required in the BOM? If so, what kind of blocks will be needed and what sections will each of them have?
 - 1.11. Are blocks of nested sections required in any section of the BOM? If so, what kind of blocks will be needed and what sections will each of them have?
 - 1.12. Which of the existing text templates will be used in the various sections? Consult about the combinations of the attribute type keys corresponding to these templates.

2. If a title block that is essential for the BOM doesn't exist, create it. A similar title block can be used as a prototype.
Title Block should contain a table to specify the BOM items to be entered. It should have the following option enabled **Use for BOM**. The table header may contain column titles. The remaining cells of the table should be empty. In the advanced format, one should select the **For BOM Table** type and the type of the input data, as well as indicate the number of the table row.
3. Create a template containing title blocks (forms) for the first and subsequent BOM pages.
4. If the required templates are missing, then create them (see Section 6.3.5 on p. 1563). Make sure you use template-prototypes.
5. To create a new style call command **Service— Libraries of Styles, Layout Types and Title Blocks— BOM Styles...** In the dialog that pops up, select the library, in which the new style will be created, and click the **Create** button.
The **BOM Style** dialog will be displayed. Along with its built-in dialogs, it allows to set up all BOM style parameters.
6. Enter a new style name and number in the library.
7. Choose a template for the first and subsequent BOM pages on the **Template** page.
8. Change the general settings on the **Settings** page.
9. Click the **Create by Default** button on the **Columns** page and modify the parameters for each column.
10. Create a list of additional columns and customize them on the **Additional Columns** page.
11. Create a list of sections and customize it on the **Sections** page. Specify the items input and sorting rules, as well as the column filling rules for each section.
12. Modify the data entry and variant information display rules on the **Variant Blocks** page, if necessary.
13. Enable additional section blocks and customize them on the **Additional Section Blocks** page.
14. Test the new BOM style: create a BOM using the new style, fill in its sections, check the link to the assembly drawing, the completeness of the additional columns, the sorting, the assignment of reference numbers, and the other parameters and settings.
15. Adjust the BOM style settings, if necessary, and test the new style again.



Do not neglect the testing of the new BOM style — it will help to avoid errors in the future when creating BOM using that style.

6.3.5. Creating custom fill-out templates

6.3.5.1. General information

Creating custom text templates makes sense if the text is generated from several fields and sorted by those fields in a manner that is different from their order (e.g., firstly — by the third field, secondly — by the first field, thirdly — by the second field).

It should also be noted, that during the creation of a BOM item, the text, which is generated by using a template, can only be placed in one column. It is recommended to place it into the column that will be used for sorting.

The fill-out template is created based on the table type attribute with a constant and equal number of rows.

An attribute column corresponds to each text (or number) field in the template (including fields for separators and constant values).

It is possible to set filling rules (type of value, range or list of values, default values) for each column (and hence, the template fields).

The field order for sorting is determined by the values of the field keys.

The field keys values should ascend according to the field sort order. They may have non-continuous numbering.

The text created from an attribute (a record) will not be sorted by the fields whose keys are zero.

With respect to the other fields, the text will be sorted in the ascending order of the field keys (at first — by the field having the smallest key, then by the field with the next key value, and so on up until to the field with the maximum key value).

In order to use an attribute type as a template for filling a column of a BOM section, the combination of attribute keys should match the combination that had been indicated at the column setup in the BOM section, and the attribute type itself should be located in the library specified at the column setup in the BOM section.

To make the text template available when creating an item in the section *Standardized Parts* of Simple and Group BOM, the attribute containing that template should be located in system library *Spc.lat* and has the keys *100, 5, 1*.

To make the text template available when creating an item in the section *Materials* of Simple and Group BOM, the attribute containing that template should be located in the system library *Spc.lat* and has the keys *100, 7, 1*.

6.3.5.2. Standard sequence of operations when creating user's text template

1. Think over what fields should form the text (record), created according to the template, and what is the order of the fields for sorting texts created according to the template.
2. Open the attribute type library.
3. Create a new attribute type in it, e.g. a table with a constant and equal number of rows.
4. Create a number of columns in the new attribute type that is equal to the required number of fields in the text created according to the template. A template field will correspond to every column of a table attribute. The template fields order will correspond to the attribute columns order.
5. Enter a name for each column (it will be displayed as a field name when editing the text in the template), column type, range of values in it and a default value.

6. For each column corresponding to the field that is a sorting parameter, set the key value indicating the sorting order.
7. Set the attribute type keys combination. According to that key combination, the created template could be selected from the attribute library and used when creating a BOM item.

6.3.5.3. Methods of creating text templates

Pay attention to some methods of creating templates.

- ▼ The invariable part of any designation (e.g. “*GOST*” or “*M*”) can be in a separate field. The various separators (dash, ratio, multiplication signs etc.) also receive their individual fields. Sorting by these fields is not available.
- ▼ The number of a standard can be divided into two parts, if sorting by the years is necessary.
- ▼ If possible, use admissible value lists and set the most prevalent one as a default value. Thus, you will speed up the value entry in the field and at the same time reduce the probability for a misprint in it.
- ▼ If a value in the field is constant (e.g. separator or GOST abbreviation), create an admissible value list, consisting of that single value. It will help to avoid mistakes due to accidental editing of the values in the field.
- ▼ If the sequencing of some row field should differ from the alphabetic one, customize it as follows. Specify a list of lines for filling this field, arrange them in the required order and enable the values sequence order registration.
- ▼ When creating new attribute types, use prototypes if possible. After creating a new attribute type from a prototype, modify its parameters: enter new column names, change the filling lists and the ranges of the values. Check if the field keys correspond to the required sorting sequence.

7. Variables, Parametrization

7.1. Operations with variables

7.1.1. General Document Variable Information

7.1.1.1. Types of variables

Use of variables in a document allows changing object parameters without direct editing of such parameters.

In KOMPAS-3D documents, the following types of variables are used.

User-defined variables — variables created by user in the Variables Window. They are used to obtain values of other variables. You can create variables of the normal type, as well as functional and interval variables (see Section 7.1.5.1 on p. 1589).

Variables of the operation parameters are variables created automatically in the process of creating numeric parameters of operations in model. The values of these variables are the values of parameters. Variables of this type can be used for managing the values of operations' parameters.

Variables of dimension are variables created by the user in the process of creation of dimensions in graphical documents or sketches. The values of these variables are the values of dimensions. If the dimension, for which a variable is created is a controlling one, the variable can be used for the managing the dimension's value. If the dimension is an informational one, it is impossible to manage its value via a variable.

Variables of maximum deviations are variables created by the user in the Variables Window. They are designed for setting the values of maximum deviations of dimensions and parameters, the values of which are expressed in linear or angular values. Work with this type of variables is described in Section 7.1.5.2 on p. 1592.

Besides that, when working with a model, variables are created automatically. These variables can be used to control exclusion of objects from calculation.

Variables of “measurement” and dimensions objects are also created automatically in the model. These objects are layout elements that are entered manually. In the first case, values of variables are measurement results; in the second case — values of dimensions for which they have been created. These variables are used to retrieve information only.

See Sections 2.9.2.1 on p. 631 and 7.2.1.10 on p. 1606 for more information about controlling and informational dimensions that are layout elements.

Variables can have different statuses (see Section 7.1.1.2 on p. 1569).

In order to perform different operations with a document's variables, the Variables Window is used (see Section 7.1.1.3 on p. 1571).

7.1.1.2. Variables Statuses

“External” and/or “informative” statuses can be assigned to variables.

You can assign the “informational” status to parameter variables of operations and dimensions. User-defined variable can be both external and informative.

If a user-defined variable has an “external” status in the document, this variable will be available if the document is inserted into another document (main document). You can change the value of the inserted external variable and use it to obtain the values of other variables of the main document.

If a user-defined variable in the document is both external and informational, then this variable will only be available for obtaining other variables' values when inserting a document into another document. You cannot change value of such variable.

For more information about informational variables see Section 7.1.1.2.1, about external variables — Section 7.1.1.2.2 on p. 1571.

7.1.1.2.1. Informational variables

Info variable is a variable the value of which depends on other variables, dimensions, positions of document objects.

A sign of an info variable is yellow or green (if it is also the external one) color of the cell with the name of the variable in the Variables Window table (see Section 7.1.1.3.1 on p. 1573).

The “informational” status can be assign to variables manually or automatically.

You can assign the status manually to user-defined variables only.

In order to assign the informational status to a user-defined variable, highlight a cell in the variable's line and call the **Informational** command from the context menu. A “checkmark” will appear next to the command name in the menu. Calling the command again will restore the initial state of the variable.

The informational status can be assign automatically to user-defined and dimensions variables in graphical documents and sketches. The status is assigned in the following cases.

- ▼ A user-defined variable can become an informational one, if an expression is set for its calculation that contains other variables. When a variable is deleted from the expression or if the whole expression is deleted, the informational status will be canceled.
- ▼ A dimension variable is assigned the informational status, if a dimension for which it is created becomes informational. If an informational dimension becomes the controlling one, the informational status of the variable will be canceled.

Besides that, when working with models, variables can be created that are informational from the beginning. In this case, it is impossible to cancel the informational status of such variable.

Informational variables in models are created for the following objects:

- ▼ dimensions — layout elements that are manually set,
- ▼ measurement objects,
- ▼ parameters of operations the values of which depend on position of model's objects (such parameters include, for example, the **Distance** parameter of the extrusion operation when extruding a sketch up to vertex, since in this case, the parameter's value depends on the vertex's position).



Note the following specifics when working with informational variables.

The method of assigning a value to a user informational variable can be modified; its respective cell of the **Expression** column in the Variables Window is available for editing.

For variables of dimensions and operations' parameters, to which the informational status is assigned, this option is not available.

7.1.1.2.2. External variables

External variables are used in parametric fragments and models.

External variable in a parametric fragment is a variable whose value is available for modification in the main document—a document in which the fragment is inserted. The main purpose of external variables in parametric fragment is controlling parameters of the fragment inserted to a document, without editing that fragment “from inside”.

An **external variable** in a model is a variable whose value is available and can be modified in another model containing this model as a component. The main purpose of external variables in a model is to control model dimensions and topology during and after inserting it into another model.

A sign of an internal variable is blue or green (if it is also the informational one) color of the cell with the name of the variable in the Variables Window table. (see Section 7.1.1.3.1 on p. 1573).

“External” status can be assigned to user-defined variables only. Variables are created and assigned with their “external” status when creating the fragment (model).

In order to assign the external status to a user-defined variable, highlight a cell in the variable's line and call the **External** command from the context menu. A “checkmark” will appear next to the command name in the menu. Calling the command again will restore the initial state of the variable.



If an external variable is also an informational one (see Section 7.1.1.2.1), it will be visible in the main document but will not be available for editing. The informational status is assigned to an external variable when creating a fragment (model).

For special details of using external variables, see Section 7.1.5.3 on p. 1593.

7.1.1.3. Variables Window

All document variables are displayed in the Variables Window (Fig. 7.1.1 and 7.1.2).

Enabling and disabling view of Variables Window is done by the command **View— Toolbar— Variables**. Also you can use the button **Variables** on the panel **Standard**.



List of variables is displayed as a table. The 1st level of the list consists of the user-created variables (they form a so called main section of the List of Variable), other levels consist of model operation parameters' values and dimension values of graphical documents and sketches. Variables of dimensions in a drawing are grouped in sections according to types of dimensions (Figure 7.1.1). In a fragment, all dimension variables form a single section. In a model, variables of operation parameters and variables of sketch dimensions are grouped in sections

in accordance to objects, to parameters of which they correspond (Fig. 7.1.2).

Name	Expression	Value	Comment
D:\Projects\Platte.cdw			
a	120.0	120.0	Part width
b	0.4*a	48.0	Part height
k	0.2*a	24.0	width of the ledge
d	10.0	10.0	bore
s	d<=10?1:1.5	1.0	width of the chamfer
h	0.1*a	12.0	thickness of the parts
m	0.5*h	6.0	height of the ledge
t	5.0	5.0	distance from the ledge to t...
l	a - k - t - d/2	86.0	distance from the end to ce...
View 1 (1:1)			
v1	b	48.0	
v2	a	120.0	
v3	k	24.0	
v4	l	86.0	
v5	d	10.0	
v6	d+2*s	12.0	
(c) View 2 (1:1)			
v7	h	12.0	
v8	m	6.0	
v9	k	24.0	
v10	l	86.0	
v11	a	120.0	
v12	d/2	5.0	
v13	s	1.0	
v14	45.0	45.0	chamfer angle

Fig. 7.1.1. Drawing Variables

Name	Expression	Value	Parameter	Comment
Branch (Bodies-1)				
s	5.0	5.0		Wall thickness
a	90.0	90.0		Angle, grad
r	60.0	60.0		Bending radius
D	50.0	50.0		External diameter
(p)Origin				
Pipe sketch				
v8		0.0	Exclude from ...	
v1	r	60.0		Bending radius
v2	D	50.0		External diameter
Revolution Operation:1				
v18		0.0	Exclude from ...	
v20	s	5.0	Wall Thicknes...	Wall thickness
v21	a	90.0	Angle 1	Angle, grad
Sketch:2				
Axis through Two Vertices:1				
Axis through Two Vertices:2				
Linear Dimension:1				
v70		0.0	Exclude from ...	
v71		85.0	Value	Overall dimension
Diameter Dimension:1				
v74		0.0	Exclude from ...	
v75		40.0	Value	Internal diameter

Fig. 7.1.2. Part Variables

The table in which the list of variables is presented contains the default set of columns positioned in a certain order. You can change the set of displayed columns and their order in the table (see Section 7.1.1.3.2 on p. 1574).



Rows of the table with variables of the main section can be moved up and down. For this purpose, the **Move up** and **Move down** commands are used that are called from the **Service** menu of the Variables window or using the respective button of the window's toolbar.

Besides that, you can sort variables of the main section based on data contained in one of the table's columns. For this purpose, highlight the cell of the column you need and call the **Service — Sort** command from the Variables Window.

7.1.1.3.1. Color indication in the Variables Window

Different colors are used for indication in the Variables Window.

Depending on the type and status of a variable, the cell with its name can have one of the following colors:

- ▼ blue— external variable,
- ▼ yellow— informative variable,
- ▼ green— external informative variable,
- ▼ pink— interval variable,
- ▼ turquoise— functional variable,

The **Tolerance** column's cell in the line of variable has light-green color, if operation's parameter or dimension, to which it corresponds, has individual tolerance (see Section 7.1.4.2 on p. 1587).

Red color is used to denote objects with errors.

7.1.1.3.2. Setup of the Variables Window

The list of variables is displayed in the form of a table in the Variables Window (see Fig. 7.1.1 on p. 1572 and Fig. 7.1.2 on p. 1573). You can control the display and mutual position of all columns of the table, except for the **Name** column.

In order to set up the set and mutual position of columns, do the following.



Call the **Service — Set up columns** command from the Variables Window. The **Set up columns** dialog will be displayed (Fig. 7.1.3).

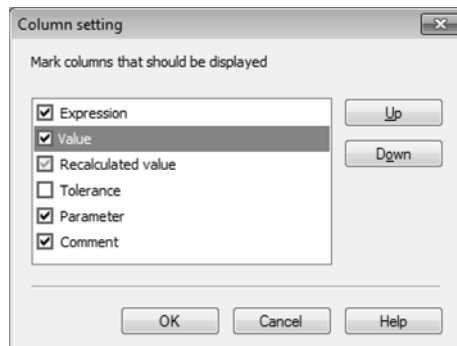


Fig. 7.1.3. Column customization dialog

The dialog contains the list of all columns of the table. You can control their display and position. An option on the left of the column's name controls its display in the Variables Window: If the option is enabled, the column is displayed in the window. Otherwise, the column is not displayed.

Include the components you need in the table by enabling and disabling options.

Note that the option that corresponds to the **Recalculated value** column is only available if the mode of recalculation of dimensions taking account of tolerances is enabled (see Section 2.10.3.3 on p. 679).

You can also modify mutual position of columns by moving their names in the dialog's list. For this purpose, the **Up** and **Down** buttons are used. Order of columns in the list defines their order in the Variables Window.

In order to finish customization of the table, close the **Set up columns** dialog by clicking **OK**.

In order to control displaying of the table's columns, apart from the dialog described above, you can also use commands of the context menu which is called in the header of any of the columns in the Variables Window. In order to enable/disable displaying of the required column, call the respective menu command. A checkmark next to the command's name means that the column's display is enabled.



The **Recalculated value** command of the context menu is only available, if the mode of dimension recalculation taking account of tolerances is enabled.

7.1.1.4. General procedure of working with variables

While working with variables, you are recommended to proceed as follows.

First of all, create user variables. These variables will be used to set values of variables of model operations' parameters and variables of dimensions of graphical documents and sketches.

When working with a graphical document, create dimension variables.

When working with a model, create variables of sketch dimensions. Variables of model operations' parameters are created automatically, once operation has been created.

Methods for creation of different types of variables are described in Section 7.1.2 on p. 1576.



Later on, you will be able to add and delete user variables and dimension variables of graphical documents and sketches. Variables of model operations' parameters are added and deleted automatically.

You can also edit variables using different methods (see Section 7.1.4.1 on p. 1585).

Use variables to control the geometry of a model or image. For this purpose, assign the required values to variables (see Section 7.1.3 on p. 1578).

As a result of assigning values to variables, dimension values in the graphical document or sketch change, as well as numeric parameters of operations in a model.

When working with a drawing, types of drawing (except for the current one) that contain dimensions with modified values are marked with a red checkmark in the Design Tree. When working with a model, icons of operations with modified parameters and sketches that contain dimensions with modified values are marked with the red checkmark in the Design Tree. This means the changes are not transferred to the drawing/model yet.



To rebuild the drawing/model, enable command **View— Rebuild**. The image will be rebuild taking into account new values of dimensions or operations' parameters.



The rebuilding command has also to be called if the status of a variables has changed; for example, if a user variables was assigned or canceled the “informational” status.

The current drawing view and image in the fragment is rebuild automatically without enabling this command, i.e. upon assigning a value for a new variable.

Also, the image in a sketch will also be automatically rebuilt if the variable's value is set when working with the sketch.

7.1.1.5. Specifics of working with variables in documents of KOMPAS-3D version 11 or earlier

KOMPAS-3D Version 9 and 12 features improved principles of operation with variables: in Version 9 — in sketches, in Version 12 — in graphic documents. Therefore, when opening a mod-

el in the current KOMPAS-3D version, which had been created in KOMPAS-3D version 8 Plus or earlier or of the graphical document, which had been created in KOMPAS-3D version 11 or earlier, the equation system of the document will be **converted**. The conversion process flows in a manner which allows you to retain “behavior” of the parametrized view (if practical), i.e. when you work in the current KOMPAS-3D version, any changes of variable values shall modify the view just as in the previous versions.

Particular issues associated with display of converted equation system in the Variables Window.

1. Variables can appear both in the first level of variables list and in groups of the next level (i.e. in one of the drawing types or among variable dimensions of a fragment/sketch). Total number of variables and expressions in a document may be changed. For example, variable b was used in two drawing views. Based on the conversion result the variables are divided by types, get various names and related expressions: variable b appears in one type, and the other type will have $b1$ which is equal to b (cell **Expression** of variable $b1$ displays the name of variable b).
2. Equations entered as assignments (i.e. equations that have in the right part numeric constants or expression containing numeric constants only) **are longer** than the area for equations. A numeric constant (or expression result) is transferred to cell **Expression** of the respective variable.
3. In a sketch created in KOMPAS-3D of version 8 Plus or earlier or in a graphical document created in KOMPAS-3D version 11 or earlier, both user variables and dimension variables can have the “external” status. If you cancel this status (through the **External** command from the context menu), you won't be able to restore it. You can assign the external status to the variables if they are user-defined only.

7.1.2. Creation of variables

7.1.2.1. User-defined variables

Do the following to create a user-defined variable.

1. Click on cell **Name** empty line of the variables list main section.
2. Enter a variable name and press *<Enter>*.

A variable name may contain letters of the Latin alphabet (case-sensitive), western numerals and the underscore characters (“_”). The length of a variable name may not exceed 512 characters. The first character in the variable name is a letter or an underscore. Examples of record: a , b_1 .

If the name format is not observed after pressing *<Enter>*, the message appears on the screen to inform that the variable name is invalid, and the name will not be saved.

3. Click on **Expression** cell of variable being created.
4. Specify a method of assigning variable value and press the *<Enter>* key. Variable value can be obtained by different methods (see Section 7.1.3 on p. 1578).

If, when filling the **Expression** cell, invalid characters were used, once *<Enter>* has been pressed, the message will appear on the screen to inform about errors, and the cell content will not be saved.



You can use mathematical expression to assign value to a variable. If the specified expression contains variables which do not exist in the document, they will automatically be created in the main section. Specify a method of assigning values to these variables.

After specifying method of assigning value to a variable, calculated value appears in the **Value** cell.

5. If required, enter comments on the created variable in cell **Comments**.

7.1.2.2. Variables of model operations' parameters

Variables of model operations' parameters are created automatically during creation of an operation. It is impossible to create this type of variables manually.

For each operation, a set of variables is created that correspond to all of its numeric parameters. The name of the variable is composed in accordance to the "vN" template, where N— is the number of the variable in the list of model's variables. Values of variables are values of parameters that are entered during creation of an operation in the fields of the Property bar. Methods of assigning values to parameters are described in Sections 2.1.4.2.1 on p. 136 and 2.1.4.2.4 on p. 140.



If an operation's parameter value depends on position of a model's object, the value created for the parameter is informational (see Section 7.1.1.2.1 on p. 1570).

For variables of each operation's parameters, a separate group is created in the Variables Window. Its name corresponds to the name of the operation. In this group, the variables of the **Exclude from calculation** parameter is also created. It controls exclusion of the operation from calculation.

7.1.2.3. Dimension variables of graphical documents and sketches

The dimension variable of a graphical document or sketch is created in the dialog for setting the dimension's value. This dialog automatically appears when dimensioning or after calling the **Set Value of Dimension** command and specifying dimension.

The dialog makes it possible to set the name of the variable, its value and enter a commentary to it. Working with the dialog is described in Section 7.2.2.20 on p. 1614.

In the Variables Window, the created variables is located as follows:

- ▼ in a model — in the section with the name of the sketch to which the respective dimension is related,
- ▼ in a drawing — in the section with the name of the view to which the respective dimension is related,
- ▼ in a fragment — all dimension variables form a single section.



The variable can be created for the associative dimension only (see Section 7.2.1.4 on p. 1602).

7.1.3. Assigning values to variables

7.1.3.1. General information

Assigning different values to variables makes it possible to control values of other variables and object parameters without direct image or model editing.

Values of user variables are used to retrieve values of other user variables, operation parameters' variables and dimension variables.

The value assignment method for a user variable is set and can be changed in the Variables Window. The following methods are available:

- ▼ directly enter the number or constant which is value of the variable (see Section 7.1.3.2),
- ▼ Enter the expression to calculate the value of variable (see Section 7.1.3.3 on p. 1579)
- ▼ Link to a variable, i.e. assigning values to another variable (see Section 7.1.3.5 on p. 1583).

Assigning different values of operation parameters' variables make it possible to control geometry of the mode; in case of dimension variables, geometry of image in a graphical document and sketch can be controlled.

Variables of model operations' parameters are assigned their initial values automatically when operations are created. Initial values of dimension variables are values of dimensions for which they have been created.

You can change the value of the operation parameter's variable or dimension variable by specifying another method of value assignment in the Variables Window. For variables of these types, the same method of value assignment as for user variables (see above) are available. Besides that, values of these variables can be modified in the dialog for setting a dimension's value or in the Property bar fields (see Section 7.1.4.1 on p. 1585).

Variables of the **Exclude from calculation** parameter are used to control exclusion of objects from calculation. By default, they have zero values. For special details of assigning values to these variables, see Section 7.1.4.1.1 on p. 1587.



Note the following specifics when choosing the method of assigning value to a variable. After each expression entering, an equation is created in a document. This equation is included into equation system of a document. The document's equation system must have a solution available for the set variable value. Otherwise, an error message appears on the screen.

In order to transfer modified values of variables to a document, it has to be rebuilt.

7.1.3.2. Entering a numerical value or constant

To assign value to a variable using this method, perform the following actions.

1. Select a variable in the Variables Window.
2. Click on the **Expression** cell of the same row.
3. Type a number, which is value of variable and press the <Enter> key.

The variable will take specified value— typed number will appear in the **Value** cell in the same row.

“name of variable = value” equation will be created in the model.

A constant can be a variable's value, as well as a number. In this case, value of the specified constant is displayed in the **Value** cell.

You can input constants from keyboard (list of constants is provided in Table 4 of Appendix IV), but it is recommended to specify them using special dialog (see Section 7.1.3.4 on p. 1582).

7.1.3.3. Entering an Expression

To assign value to a variable using this method, perform the following actions.

1. Select a variable row in the Variables Window.
2. Click on the **Expression** cell of the same row.
3. Type the expression to calculate the value of variable and press <Enter> key. An expression can be input from the keyboard or inserted from the special dialog (see Section 7.1.3.4 on p. 1582). Syntax of expressions is described in detail in Section 7.1.3.3.1.

Examples of using an expressions to obtain variables values of model are provided in Sections 7.1.3.3.2 on p. 1581 and 7.1.3.3.3 on p. 1581.

After entering the expression, variable value will be automatically calculated in accordance with the current values of variables used in it. Calculation result will appear in the **Value** cell.

“name of variable = value” equation will be created in the document. An equation may contain maximum 80 elements — numbers, constants, variables, signs of operations and functions. If this number is exceeded, the screen will show a message that the expression is too complex.

Upon entering each new equation, the output system of equations is automatically checked for circular references and for variable self-assignments. The message will popup if one of the specified errors take place.

Circular reference **is a reciprocal dependence of two equations**. A simple example of a circular reference— a system of equations as follows:

$$\begin{aligned} a &= b + 1, \\ b &= a + 1. \end{aligned}$$

Circular reference of the model variables can be realized by the hierarchy of objects relations in the model.

For example, a **sketch 1** is created in the model with the X variable. On the basis of this sketch, an extrusion element is created on its face (**sketch 2**) with the Y variable. X and Y variables are related by the equation (Fig. 7.1.4).

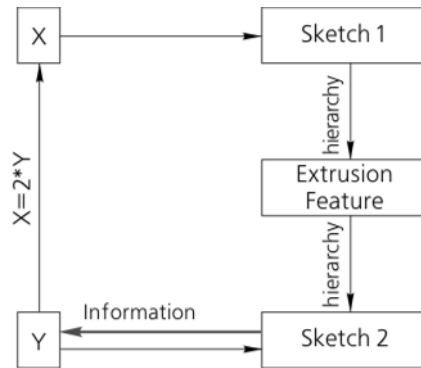


Fig. 7.1.4. Occurrence of circular reference

Until the variable Y is informational, there is no circular reference. Should you edit the sketch to turn variable Y to informational one (to do this, it is sufficient to delete the constraint *fixed* for the respective dimension), then in the model system of equation a circular reference shall appear: to rebuild **Sketch 1**, it is essential to know the value of variable X , to do this, you should know the value of variable Y , which can be defined only after rebuilding of **Sketch 2**, and this in turn shall require rebuilding of the extrusion element and **Sketch 1**.

7.1.3.3.1. Syntax of expressions

You can use the following elements when making expression.

- ▼ Arithmetical operators (except for “=” operator), logical operators and functions.
- ▼ Numerical values and constants. Example of using constants: “a - sin (b * M_PI) <= FLT_EPS ? c : d”.

The full list of operators, functions and constants is provided in Appendix IV.

- ▼ Any variables contained in the document, except for interval ones (see Section 7.1.5.1.2 on p. 1591).



If a name of a variable that is not present in the document is used when creating an expression, user value with the same name will be created after the expression is fixed. This variable is placed in the main section of the Variables Window.

- ▼ Formulas, the list of which can be found in Section III.



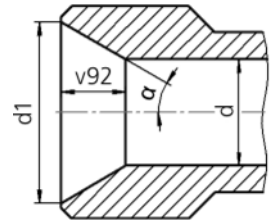
An expression (except for the variable name and the numerical value) can include any number of blank spaces. Those are ignored when interpreting the expression.

Examples of using an expressions to obtain variables values of model are provided in Sections 7.1.3.3.2 and 7.1.3.3.3.

7.1.3.3.2. Example of using an algebraic expression

The sketch of extruded element is the circle whose diameter is $d1$. The element forms conical part of the hole. The diameter of cylindrical part of the hole is d (Fig. 7.1.5, b).

Variables				
Name	Expression	Value	Parameter	Comment
Part (Bodies-1)				
d	10.0	10.0		Inner diameter
d1	17.0	17.0		Diameter of inner cone d1
right	0.0	0.0		right side of the crossing
a	30.0	30.0		The angle of the inner cone
(p)Origin				
Sketch:1				
Rotation:1				
Sketch:3				
v76	right	0.0	Exclude from Calculation	right side of the crossing
v30_d1	d1	17.0		Diameter of inner cone d1
Inner cone				
v86	right	0.0	Exclude from Calculation	right side of the crossing
v92	$(d1-d)/2*\tan(a/2)$	6.0622	Distance 1	
v94	a	30.0	Angle 1	The angle of the inner cone



a)

b)

Fig. 7.1.5. Algebraic expression

Extrusion depth ($v92$) should be calculated by formula (Fig. 7.1.5, a):

$$(d1-d)/2*\tan(a/2),$$

where:

$d, d1$ — diameters of cone bases,

a — semicone angle.

7.1.3.3.3. Example of using a logical expression

Extrude element, forming the shoulder, should be presented in the model of flanged elbow if one of the following conditions is satisfied:

- ▼ GOST 20189, angle 90°
- ▼ GOST 20190.

The following variables should be created in the model to satisfy this requirement (Fig. 7.1.6):

- ▼ setting number of the standard, *GOST*, for example
- ▼ setting angle value, *angle*, for example.

Name	Expression	Value	Parameter	Comment
GOST	20189.0	20189.0		
angle	90.0	90.0		
h_shoulder	2.50	2.50		
Ledge				
v86	GOST==20189&&angle=...	0.0	Exclude from Calculation	
v89	h_shoulder	2.50	Distance 1	
v91		0.0	Angle 1	

Fig. 7.1.6. Logical expression

Expression to calculate value of shoulder variable, corresponding to **Exclude from Calculation** parameter (*v86*), should be the following:

$GOST == 20189 \&\& angle == 90 || GOST == 20190 ? 0 : 1$ (Fig. 7.1.6).



Spaces and brackets can be used in expressions for your convenience.

$(GOST == 20189) \&\& (angle == 90) || (GOST == 20190) ? 0 : 1$

7.1.3.4. Insert Expression Elements

You can insert system functions, constants, operators, whole expressions and curve functions as expression elements.



Expression element inserting is performed using **Insert Mathematical Expression** dialog. To call the dialog, press the **Insert Function** or **Insert Constant** button on the Variables Window toolbar. These buttons are available when editing **Expression** cell contents.



The dialog contains list of expression elements available for insertion. They are grouped. To select the required element, expand the group in which it is contained. List of this group elements will be displayed as a table in the right part of the dialog.

In the lower part of the dialog, you can see the **Composition of equation** field. It is used to display the resulting value—the element as it will be transferred to the Variables Window. The field is filled in automatically after selecting an element. Field contents are saved until closing the dialog. When reselecting element, field contents are changed.

If expression is selected for inserting, a list of variables included into this expression is displayed in a table view in the dialog. This table allows to assign arbitrary names to variables.

To assign a name to a variable, click the mouse in a cell of the **Name insertion** column, enter the required name and press **<Enter>**. This name will be used to denote a variable in the resulting expression displayed in the **Composition of equation** field.

If necessary, you can edit the field contents. To do this, enable an option corresponding to it.

When editing, the following actions are available.

- ▼ Entering symbols from the keyboard.
- ▼ Inserting symbols using the clipboard.
- ▼ Adding expressions and their elements from the element table of the current dialog. The selected element will be added to the place of the resulting expression where the cursor is posi-

tioned at the moment of adding. If fragment of the resulting expression is highlighted at the moment, it is replaced with the added element.

- ▼ Inserting variables used in the document. It is performed by using the **Select variable** command of the context menu of the **Composition of equation** field. After calling this command, **Variables** dialog allowing to select the required variable from the variables list of the current document appears on the screen.

Edit the resulting expression using the actions described above.

You can save the resulting expression to the file of mathematical expressions *collection.law*. The file is created automatically when the first expression is saved. The file path is defined by the `EXPRCOLLECTION` variable of the environment KOMPAS-3D (see Section 9.5.4 on p. 1886). Use *Kompas.ini* (see Section 9.5.3 on p. 1883) if the name and/or position of the file of collection of mathematical expressions should be different from the default values.

In order to save the resulting expression to the file, call the **Save expression** command from the context menu.

Expressions saved in the collection file are automatically added to the **From collection file** group of the **Insertion of mathematical expression** dialog, and can be inserted to the Variables Window. Operations with expressions from the collection file are the same as operations with expressions and their elements contained in other dialog groups.

To insert resulting expression into Variables Window, press **OK** button. **Insert Mathematical Expression** dialog will close, generated element will appear in the **Expression** cell of the variable being edited.



Note the following specifics when inserting curve functions.

Curve functions are presented in the dialog by functions' expressions for X, Y, Z coordinates. Functions' expressions should be inserted separately for each coordinate.

For example, to insert a circle evolvent expression, you should first insert the expression for calculation of X coordinate by highlighting the respective line and clicking **OK**, and then re-open the dialog and insert the expression for calculation of Y coordinate.

When highlighting a line with the name of a function (in the example mentioned— **Circle evolvent**), the resulting expression will also include expressions for calculation of all coordinates and additional characters. Such expression cannot be used for obtaining the variable value. It is required to edit it.

7.1.3.5. Link to a Variable

As a link source document, you can use a graphical document or a model.

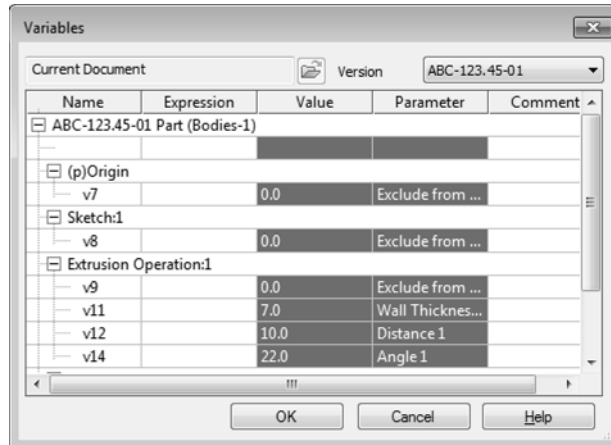
To assign value to a variable using this method, perform the following actions.

In the Variables Window select a variable which is to take a value through the link, click in any cell of this variable line.



Click **Insert Link** on the Toolbar of the Variables Window.

The **Variables** dialog will appear (Fig. 7.1.7). In the dialog, the full name of the current document and the table of variables contained in it are displayed.

Fig. 7.1.7. **Variables** Dialog

In order to create a link to the current document's variable, specify the required variables in the dialog's table.

If the current document is a model with versions, you can create a link to the variable of any of the versions. In this case, the **Version** drop-down list is available which contains designations of all versions that are present in the model. After selection of the required version from the list, the dialog will display the table of the version's variables. Specify the required variable in the table.



If you need to create a link to another document's variable, click the **Source file** button and specify the document you need. If a model with version is chosen as the source document, the drop-down list of versions' designations will be displayed in the right part of the dialog. Choose the designation of the required version in the list.

Once the document is chosen, the **Variables** dialog with the table of the document's variables will appear on the screen again. Specify the required variable.

To finish the variable selection, click the **OK** button.

The variable specified in the Variables Window will get the value of the variable which is the source of the link. It will be displayed in the **Value** column.

"name of variable = value" equation will be created in the model.

In the **Expression** column, text of the link to the variable will appear (see Fig.7.1.8). Link text is composed according to the following template:

<full source file path>|<version number (for models)>|<variable name>

For example, a link is created to the *H* variable in the drawing *C:\Work\Project.cdw*.

Link text will be the following: *C:\Work\Project.cdw|H*.



If another version of the current model serves as the source of the variable the full source file path is not included in the link text.

7.1.3.6. Updating Links to Variables

When value of the variable (on which link from other document points to) changes in the source file (see Section 7.1.3.5), it is necessary to transmit new value of the parameter into this document— update a link.

It is possible to update links if the variable source file is open. If a model is a variable source, open the model which contains the source as a component.



To quickly open the link source file, enable command **Open Source File** from the context menu of the variable reference (Fig. 7.1.8).

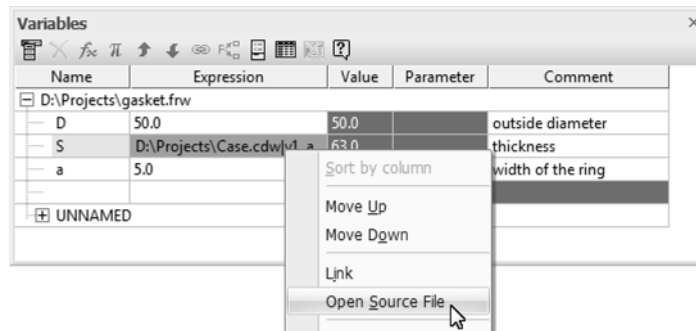


Fig. 7.1.8. Variable reference



Once you have open the source file, enable command **View — Rebuild** to obtain new values of variables.

The source file is not rebuilt while rebuilding the document using links.

While opening a document with a variable link, the system checks whether the source is open. If the source is closed, link-variable remains highlighted red in the Variables window until it is properly updated.

7.1.4. Editing and deleting variables

7.1.4.1. Editing variables

Variables are edited in the Variables Window. As part of editing, you can modify the following parameters of variables:

- ▼ method of assigning values to variables — for user variables, dimension variables of graphical documents and sketches and variables of model operations' parameters (including the **Exclude from calculation** parameters).
- ▼ commentary to a variable — for all variables.



If a variable has the informational status, you can only edit its commentary. The exception are informational user variables. Changing the method of value assignment remains available for these variables.

In order to set and modify the method of assigning the variable's value, the **Expression** cell is used (all supported methods of value assignment are described in Section 7.1.3 on p. 1578).

By changing values of model's variables, the following should be noted.

- ▼ Different parameters of operations use different value ranges. For example, the value of the variable assigned to the **Angle** parameter (this parameter is used by revolved, extruded and other features) cannot be less than zero or greater than 360. Sometimes, it happens so that parameters, though having different value ranges, are assigned to one and the same variable. The variable can later be assigned a value outside the range set for one of the parameters. In this case, an error appears in the model.
- ▼ If the variable of the **Exclude from Calculation** parameter has an assigned expression or link, then a random exclusion of the corresponding object from calculation becomes impossible (the command **Exclude from Calculation** in the object context menu becomes inaccessible). For variable dimensions of graphical documents and sketches and model variables, there are further editing options.

When working with a graphical document or sketch, in order to edit the dimension variable, you can use the dialog for setting the value of the dimension which appears after double-clicking its dimension text. The dialog makes it possible to change not only the variable value assignment but also the variable's name. Working with the dialog is described in Section 7.2.2.20 on p. 1614.



In case of the informational variable, i.e. the variable is related to the informational dimension, you can only change its name and commentary in the dialog of the dimension value assignment (changing the value assignment method is not possible).

When working with a model, you can modify the method of value assignment to an operation's parameter variable by changing the value of the numeric parameter that corresponds to it in the Property bar. Method of value assignment in the Property bar fields are described in Sections 2.1.4.2.1 on p. 136 and 2.1.4.2.4 on p. 140.

If a parameter's value is expressed in linear or angular values, in order to edit variables, you can also use the dimension value assignment dialog, which appears after double-clicking the dimension text of the operation's dimension that corresponds to the required parameter. The dialog makes it possible to modify the method of the variable's value assignment or its comment. Working with the dialog is described in Section 2.9.2.2.1 on p. 634.

Besides that, the dialog is used for editing dimension variables that are manually set layout elements. In the dialog, you can change the name and commentary for the variables.



You can modify the sketch dimension variable when editing a 3D object that uses the sketch. For this purpose, run the object edit mode. In the model window, dimensions of the sketch in use will be displayed. Call the dialog for setting the required dimension's value by double-clicking its dimension text and perform the required operations with respect to modification of variable. In the dialog, you can change the variable's name, method of its value assignment and its commentary.

7.1.4.1.1. Peculiarities of variables editing excluding objects from calculation

The **Exclude from calculation** parameter's value is created with zero value by default. It means that the object to which it corresponds, is included in the calculation. In order to exclude an object from calculation, value of the variable should be equal to one.

In order to exclude an object from calculation, enter any number other than zero, a constant or expression in the **Expression** cell of its respective variable. The result of the expression's calculation should be other than zero. In this case, the variable's value will automatically become equal to one, and the object will be excluded from calculation. Note that the entered value or constant is saved in the **Expression** cell, while the entered number is replaced by *1.0*.

The variable of the **Exclude from calculation** parameter automatically gets the value equal to one in the following cases:

- ▼ if the object is excluded from calculation using the **Exclude from calculation** command of the context menu,
- ▼ if the object's property create from the variable of the **Exclude from calculation** parameter (see Section 5.1.3.2.1 on p. 1383) has the **Enabled** value.

In these cases, after inclusion of the object in calculation (using the respective command or by assigning the **Disabled** value to the property), the value of the variable will again become equal to zero.



If an object is excluded from calculation as a result of manual entry in the **Expression** cell of the Variables Window, the object can be included in calculation only by manual entry of the zero value in that cell. The command for inclusion of the object in calculation and changing the value of the respective property are not available.

If the object should be excluded from calculation when a certain condition is met, a logical expression is used for setting the variable's value. Example of using such an expression can be found in Section 7.1.3.3.2 on p. 1581.

7.1.4.2. Assigning tolerances in the Variables Window

Using variables makes it possible to control the values of numeric parameters of models' operation parameters and controlling dimensions of a graphical document or sketch. Values are managed in the Variables Window. Besides that, you can use this window to assign or change tolerance for a parameter's value or tolerance for a dimension.

Tolerances are displayed in the **Tolerance** column. By default, this column is not displayed in the Variables Window. In order to display a column, perform steps described in Section 7.1.1.3.2 on p. 1574.

If a cell in the **Tolerance** column of the variable's line is filled, you can view the cell, set and change tolerance. The cell is filled in the following cases:

- ▼ when working with a model:
 - ▼ if the variable corresponds to parameter of the operation, the value of which is expressed in linear or angular values.
 - ▼ if the variable corresponds to the controlling dimension of a sketch,
- ▼ when working with a graphical document, if the variable corresponds to a controlling dimension.

In all other cases, the cell of the **Tolerance** column in the variable's line will remain empty.

When working with a model, the tolerance of a parameter value or a controlling dimension (in a sketch) can be either general or individual (see Section 2.10.2 on p. 667). All tolerances are individual in graphic documents. Both general and individual tolerances are displayed in the Variables Window.

The tolerance is a range between limit values of a parameter or dimension. For example, the class of accuracy tolerance of *H10* with limit deviation values $+0,00$ and $+0,12$ can be assigned to a linear dimension *60* with a variable. This tolerance is displayed in the Variables Window as $[60;60,12]$.

If a value of a parameter or dimension has no individual or general tolerance assigned, limit values of the range are the same as the nominal value of the parameter (dimension). For example, no tolerance is assigned to linear dimension *60* with a variable. Contents of the cell in the **Tolerance** column in the Variables Window is $[60;60]$. Such record is used if the **Nominal** option was enabled during the individual tolerance assignment.

Cells of the **Tolerance** column are gray, except for the cells containing values of individual tolerances. These cells are light-green. Irrespective of color all cells of the **Tolerance** column are not available for manual entry. Call the tolerance assignment dialog through one of the following methods to assign or modify a tolerance:

- ▼ highlight the cell with the required variable and call the **Service — Assign tolerance** command in the menu of the Variables Window.
- ▼ calling the **Assign Tolerance** command from the context menu of the variable's name
- ▼ double-clicking the cell of the **Tolerance** column with the respective variable.

The tolerance assignment dialog is illustrated in Fig. 2.10.2 on p. 669. Working with the dialog is described in Section 2.10.2.2 on p. 669.



An empty cell in the **Tolerance** column might also suggest that the variable matches the dimension or parameter that the use of tolerance is disabled for. In this event, you can call the tolerance assignment dialog to enable the use of tolerance and set its value.

7.1.4.3. Deleting variables

You can remove a variable in different ways.



In order to delete a user variable, select the cell with its name and click **Delete** on the Window's toolbar or press <Delete> key. The variable is deleted if its not part of expressions set for calculation of other variables. Otherwise, the variable will not be deleted, and the **Use of Variable** dialog appears on the screen with the list of the current document's objects in which the variables is used.

A variable related with dimension of graphic document or sketch can be deleted either by:

- ▼ deleting the variable's name in the dimension setting dialog (see Section 7.2.2.20 on p. 1614),
- ▼ or deleting the *Dimension with Variable* constraint (see Section 7.2.3.1 on p. 1618).

If the deleted dimension variable is included in expressions set for calculation of other variables, it is included in the number of user variables. The current value of the variable or expression set for its value calculation will be saved. In the latter case, the variable will be automatically assigned the informational status.

Variables of model operation parameters, dimensions that are manually set layout elements and measurement objects are deleted automatically when removing the respective parameters/dimensions/measurement objects. Arbitrary deletion is not possible.



If necessary, you can delete not the variable itself, but the expression set for calculation of its value. For this purpose, select the **Expression** column's cell in the variable's line and click the **Delete** button or the <Delete> key. The expression will be deleted. The variable will keep its current value.

7.1.5. Extra options for working with variables

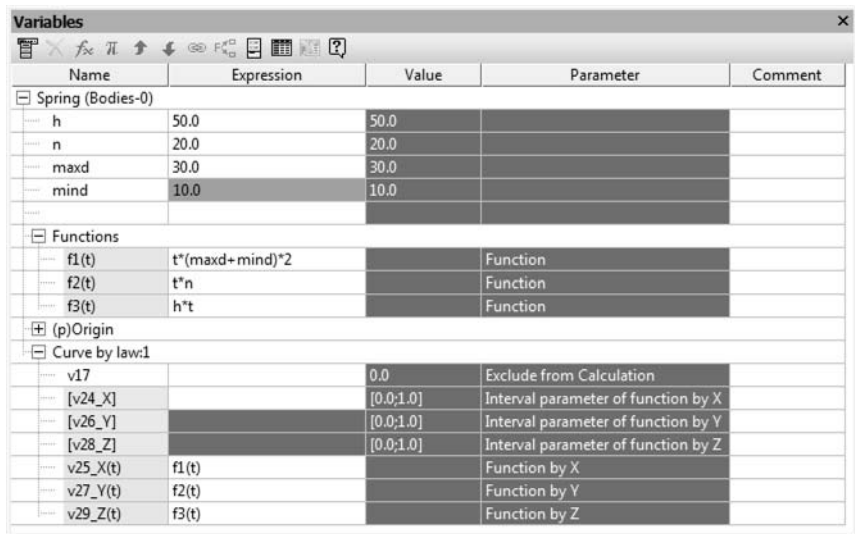
7.1.5.1. Functional and interval variables

KOMPAS-3D supports additional options of working with the variables — creation of functional and interval variables. These variables are used for obtaining values of object construction parameters. For example, functional variables can be used when creating curves by law (see Section 2.5.4.12 on p. 400), as well as when it is necessary to obtain the value of the function using different argument values (see Fig. 7.1.10). Functional variable can be also used to calculate values of user-defined variables.

Functional and interval user variables are set in the main section and automatically inserted into special sections of the variables list. functional — to the **Functions** section 7.1.9(Fig. 7.1.10 and), interval — to the section **Intervals**. These sections follow the main section.

The **Functions** section is created automatically after entering the name of the first functional variable, section **Intervals**— of the first interval variable.

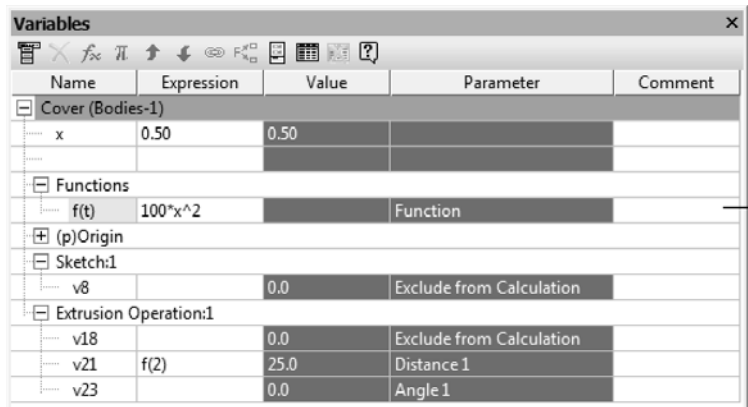
Description of functional variables can be found in section 7.1.5.1.1, interval— in section 7.1.5.1.2.



The screenshot shows the 'Variables' window with a table of variables. A bracket on the right side groups the 'Functions' section as 'Functional variable'.

Name	Expression	Value	Parameter	Comment
Spring (Bodies-0)				
h	50.0	50.0		
n	20.0	20.0		
maxd	30.0	30.0		
mind	10.0	10.0		
Functions				
f1(t)	$t*(maxd+mind)*2$		Function	
f2(t)	$t*n$		Function	
f3(t)	$h*t$		Function	
(p)Origin				
Curve by law:1				
v17		0.0	Exclude from Calculation	
[v24_X]		[0.0;1.0]	Interval parameter of function by X	
[v26_Y]		[0.0;1.0]	Interval parameter of function by Y	
[v28_Z]		[0.0;1.0]	Interval parameter of function by Z	
v25_X(t)	f1(t)		Function by X	
v27_Y(t)	f2(t)		Function by Y	
v29_Z(t)	f3(t)		Function by Z	

Fig. 7.1.9. Using functional variables in a curve by law



The screenshot shows the 'Variables' window with a table of variables. A bracket on the right side groups the 'Functions' section as 'Functional variable'.

Name	Expression	Value	Parameter	Comment
Cover (Bodies-1)				
x	0.50	0.50		
Functions				
f(t)	$100*x^2$		Function	
(p)Origin				
Sketch:1				
v8		0.0	Exclude from Calculation	
Extrusion Operation:1				
v18		0.0	Exclude from Calculation	
v21	f(2)	25.0	Distance 1	
v23		0.0	Angle 1	

Fig. 7.1.10. Functional variable usage for different argument values



When creating a curve by law, functional and interval variables are created automatically when specifying functions of curve points' coordinates and interval parameters of these functions. The created variables are not added to the **Functions** and **Intervals** sections of the Variables Window. They are placed in the section of variables of curves by law.

If necessary, you can use previously created functional and interval variables when creating a curve (see Sections 2.5.4.12.2 on p. 402 and 2.5.4.12.3 on p. 405).

7.1.5.1.1. Functional variables

Functional variable is a variable of **a(b)** type.

Name of functional variable is recorded in $f(x_1; x_2; \dots; x_n)$ type. Designations of functions and arguments may contain letters of the Latin alphabet (case-sensitive), western numerals and the underscore characters (“_”). The length of a variable name may not exceed 512 characters. The first character in the designation of function or argument — a letter or an underscore. Examples of record: $a(b)$, $a(b_1; b_2)$. If required, you can create several functional variables with the same arguments.



Name of functional variable argument can coincide with the name of normal type user-defined variable. For example, a document may contain variable b and functional variable $a(b)$. The variable and the argument with the same name are individual values with their own values, different from each other.

In the **Expression** cell of the Variables Window, that corresponds to the functional variable, the expression is entered (see section 7.1.3.3 on p. 1579) for calculation of the function's value when inserting values of its arguments.

Functional variable do not have the current value.

To use the value of the function with a certain argument value to calculate another variable or as a threshold of the interval, the functional variable should be written with an argument— a number or a constant.

For example, we have functional variable $a(b)$. You need to use value of this variable calculated for argument that is equal to 10 in the expression. To do this, write the functional variable as $a(10)$ in the expression.

7.1.5.1.2. Interval variables

Interval variable— variable of $[a]$ type.

A variable name is enclosed in square brackets and may contain letters of the Latin alphabet (case-sensitive), western numerals and the underscore characters (“_”). The length of a variable name may not exceed 512 characters. The first symbol in the variable name is a bracket, then goes a letter or underlining, and the last symbol is a bracket. Examples of record: $[a]$, $[b_1]$.

The interval variable value is an interval of values, for example, $[10.0; 20.0]$.

To assign a value to the interval variable do the following.

1. Select a variable row in the Variables Window for which you need to enter the range of values.
2. Click on the **Expression** cell of the same row.
3. Input the required value range and press the <Enter> key. Specified range is entered in brackets. The values limiting the interval are separated by “;” sign. Use the following as thresholds for the interval:
 - ▼ numerical values, for example, $\llbracket 0.0; 10.0 \rrbracket$, $\llbracket 10; -10 \rrbracket$,
 - ▼ constants (the list of constants is provided in Table 4 of Appendix IV), for example, $\llbracket 0.0; M_{PI} \rrbracket$,
 - ▼ expression (see Section 7.1.3.3.1 on p. 1580), for example, $\llbracket a-b; a+b \rrbracket$,
 - ▼ any variables, except for interval ones, for example: $[0.0; a]$, $[a(10); a(20)]$.

When setting the interval of values, the second values can be less than the first one.

After entering the range of values, the current range of variable values is calculated in **Expression** cell. It is displayed in **Value** cell and contains only numerical values.



If specified value range contains variables which do not exist in the document, they will be automatically created in the main section or in the **Functions** group (if functional variables are used). Specify values or expressions for these variables.

7.1.5.2. Variables of limit deviations

Variables of limit deviations are designed for setting values of limit deviations for controlling dimensions of a graphical document or sketch and parameters of model's operations whose values are expressed in linear or angular values.

Variables of this type are set by the user. In order to create them, choose the *initial variable*, which is a variable of dimension or parameter, the limit deviations of which should be set by the variables.



In order to create variables of limit deviations for a dimension or parameter, select the cell with the name of its respective variable and call the **Insert — Limit deviations** command from the menu of the Variables Window.

Two variables are created in the model: the variable of the upper and the variables of the lower limit deviation. They are positioned immediately after the initial variables and are hierarchically subordinate to it.

Name of limit deviation variables are formed automatically and look as follows:

- ▼ of a upper limit variable — *<name of the initial variable>_ES* (for example, *v7_ES*).
- ▼ of a lower limit variable — *<name of the initial variable>_EI* (for example, *v7_EI*).

If a variable with the same name already exists, a serial number is added to the name of the newly created variable, for example: *v7_ES_1*.

By default, variables of limit deviations get values of limit deviations of the dimension or parameter for which the have been created. These values may correspond to a common or individual tolerance for dimension (parameter); they can also be equal to zero, if no tolerance is assigned. Values are placed in **Value** cells of limit deviation variables, while their respective **Expression** cells remain empty. In order to set the required values of variables, enter the following in the **Expression** cells:

- ▼ number or constant (see Section 7.1.3.2).
- ▼ expression to calculate the value (see Section 7.1.3.3 on p. 1579),
- ▼ link to variable (see Section 7.1.3.5 on p. 1583).



If the **Expression** cell of at least one limit deviation variable is filled, the tolerance for the respective dimension (parameter) can be assigned using these variables only. Assigning tolerance is not available in the **Dimension text entry** dialog (see Section 3.3.1.3 on p. 979), **Assign tolerance** (see Section 2.10.2.2 on p. 669) and **Value and tolerance** (see Section 2.13.3.5.4 on p. 768).

If necessary, you can delete variables of limit deviations. They are deleted in the same way as user variables are deleted. Besides that, these variables are deleted automatically when their

initial variables is deleted. Specific aspects of deleting variables are described in Section 7.1.4.3 on p. 1588.

7.1.5.3. External variables

As a rule, some variables in a parametric image or in model are independent (their values may be entered by user directly), the others are computed (their values depend on values of other parameters). Independent variable values should usually be specified by the user when inserting a parametric fragment into another document or when inserting a model into another model. Other variables are computed according to the expressions existing in the fragment or model being inserted.

For example, you created an image of a cap in a parametric fragment, and then defined relations between its height, thickness, diameter and its inner hole diameters using expressions.

When you are inserting the fragment containing the cap into a drawing, the only dimensions that drive all its parameters should be the diameter and the height. Therefore, make external variables out of the cap diameter and height. Then, only values of such variables have to be specified upon inserting the fragment into the drawing. Computing values of other variables and corresponding image rebuilding will be done automatically.



A table of variables can be used for quick assignment of predetermined values to external variables of the fragment or model being inserted (or that has already been inserted) (see Section 7.1.5.4 on p. 1594).

7.1.5.3.1. Variables of parametric fragments

When inserting parametric fragment into another document, external variables of the fragment keep their names and current values. Sequence of fragment insertion is described in Section 3.7.2.1 on p. 1248.

Later, when working with the document, you can change the values of external variables of the inserted fragment.

Explode command allows to split inserted fragment into separate objects. At the same time, external variables of the fragment will be lost, while variables related to the fragment's object dimensions will be added to the group of variables of the document. They will automatically obtain names generated as per the template: "vN_name", whereas N — sequence number of the variable in the document variables list, a name— name of the fragment's external variable. The current variable values will be transferred from the fragment and displayed in **Value** cell of Variables Window. **Expression** cells will remain empty

7.1.5.3.2. Variables of models

When inserting a model into another model as a component, all its external variables are placed into this component group of variables. They automatically obtain names generated as per the template: "vN_name" (Fig. 7.1.11), where:

- ▼ N is the sequential number of the variable in the variables list of the model that contains the component,
- ▼ 'name' is the name of the component's external variable.

Cells **Value** of the obtained variables contain current values of their respective external variables transferred from the source file. **Expression** cells are empty.

Name	Expression	Value	Parameter	Comment
(+ Assembly (Tel-0, Units-0, Parts-1))				
L1	50.0	50.0		The length of branch pipe
(p)Origin				
(f) T-branch				
v8		0.0	Exclude from Calculation	
v9		1.0	Fix Component	
v8_Di		50.0		The diameter of the outer
v8_Dp		50.0		The diameter of the outer
v8_Li	L1	50.0		The length of branch pipe
v8_Lp		100.0		The length of pipe
v8_si		5.0		Pipe wall thickness
v8_sp		5.0		Branch pipe wall thickness

Fig. 7.1.11. Assembly Variables

Until the component external variable is not preset manually in the model, such variable remains related with the component source file, i.e., the component is re-constructed in the model that contains it after the variable value is changed in the source file.

If a value or expression for the component external variable has been input in the model, the relation between the variable and the source file no longer exists. Should any changes be made to the model that contains the component, the variable will have a value assigned to it in the assembly.

To restore relation between the component external variable and the source file, enable the **Value from the Source** command from the variable context menu.

Variables are created and assigned with the “external” status in the source file.

7.1.5.4. Tables of variables

Table of external variables values (table of variables) is a table stored in a file and containing predetermined values of external variables of that file.

Such a table is used when inserting a file with external variables into another document. To do this, select a row of the table. Each cell of such a row contains a value of a single external variable. These values are assigned to external variables of the file being inserted.



It is not necessary to assign variable values from a table. You can specify arbitrary values of external variables manually. If a table of variables is missing in a file, then manual entry of variables values is the only method of their modification.

The user generates a table of variables when creating or editing the file that contains external variables. A table can be edited or deleted from the file in the future.

A table of variables is organized in the following way (Fig.7.1.12):

- ▼ Starting from the second cell, the first row contains variables names, i.e. table columns headers.

- ▼ Starting from the second cell, the first column contains comments to the rows.
- ▼ Other cells contain variables values.

Thus, every table cell starting from the second one, contains a definite set of variables values and a comment, i.e., the name of this set.

Comment	H	D	L
Variant 1	15	10	40
Variant 2	15	12	50
Variant 3	16	16	60

Fig. 7.1.12. Table of variables example



Full coincidence of variables and table within a file is not necessary. When selecting a row of a table, values will be assigned only to file variables whose names correspond to table column headers.

One may use the external variables reading function when creating a table of variables in a file that contains external variables. As a result, columns corresponding to external variables existing in a file will be automatically created in a table.

Table of variables saved in the file can be exported to Excel file. Importing from Excel tables is also possible. For the reading to be available, Excel file should meet certain requirements (see Section 7.1.5.4.3 on p. 1598).

When creating or editing a table, the user may select the default method of displaying a table at inserting the file into another document: display the entire table or the first column (i.e., row comments only).

7.1.5.4.1. Creating, Editing and Deleting a Table of Variables

To create a table of variables in the current document, do the following:



1. Open Variables Window



2. On the Variables Window Toolbar (see Fig. 7.1.1 on p. 1572), press the **Table of Variables** button.

The **Table of Variables** window will pop up (Fig. 7.1.13). A newly created table of variables is empty.

3. Create a table.



- 3.1. If the required table has been created beforehand and saved in Excel file, then press the **Read from File** button located on the Toolbar of the **Table of Variables** window. Select the required file and press the **Open** button in the open file dialog. If the selected file meets the requirements (see Section 7.1.5.4.3 on p. 1598), then its data will be used to fill the table.



When you transfer data from the *.xls file to KOMPAS-3D, we do not recommend to open the *.xls file in MS Excel.

- 3.2. Press the **Read External Variables** button if the file already contains external variables. Columns corresponding to the existing external variables of the file will appear in the table.



Automatic variables tabulation excludes informational variables.

- 3.3. Use the following buttons to add the required number of rows and columns into the table:



- ▼ **Insert Row above**



- ▼ **Insert Row below**



- ▼ **Insert Column to the Right**



- ▼ **Insert Column to the Left**

- 3.4. Add comments to the rows and variables values.

4. Press **OK** to close the **Table of Variables** window.

Comment	z	d	D	b	d1	a	c	r
Light series, Z=6	6	23	26	6	22.1	3.54	0.3	0.2
Light series, Z=8	8	32	36	6	30.4	2.71	0.4	0.3
Middle series, Z=8	8	36	42	7	33.5	1.02	0.4	0.3
Middle series, Z=10	10	82	92	12	77.1	3	0.5	0.5

Fig. 7.1.13. **Table of Variables** Window

The created table will be added into the current file.

Editing a table of variables is practically identical to its creating. To start editing, open the **Table of Variables** window by performing p. 1 and p. 2 of the operations sequence described above. The following capabilities of variables table editing are available.



- ▼ Replacement with a table saved in Excel format. To do this, click the **Read from File** button.



- ▼ Adding columns corresponding to external variables of a file. For that, press the **Read External Variables** button. The table will be completed with the external variables of the file which the table does not contain yet.



- ▼ Renaming of columns. For that, select the required column and press the **Rename Column** button.

- ▼ Adding the rows above and below the selected row.

- ▼ Adding the columns to the left or to the right of the selected column.



▼ Deleting selected rows.



▼ Deleting selected columns.



▼ Moving the selected row up.



To edit a comment or variable value, click the required cell to activate it and enter a new text or number.



A table of variables may be deleted if it is no longer required. For this purpose, click the **Delete Table of Variables** button on the Variables Window Toolbar. Confirm deleting the table of variables upon system request.

7.1.5.4.2. Table of Variables Using

The main function of table of variables is quick assignment of values to external variables of the file being inserted (or which has already been inserted) into another document.

To assign values to external variables of a file, open a table of variables that this file contains and select the required row. Values of variables will be sent from cells of that row into the file.



To open the table of variables, click the **Table of Variables** button. It can be found on the Property Toolbar:

- ▼ when working with models (adding a component from a disk or model library, editing the model added from model library) — in the **Parameters** tab.
- ▼ When working with graphical documents (adding a fragment from a disk or fragment library, editing such fragment)— in the page **Variables**.

When a component was added to a model from a disk, the **Table of Variables** command is used to open the table of variables of such component. This command can be found in the context menu of that component icon in the Design Tree.



The **Table of Variables** button (command) is available if the table of variables of the file being inserted (or which has already been inserted) contains more than one row.

The **Table of Variables** window pops up when pressing the **Table of Variables** button or calling the **Table of Variables** command (see Fig.7.1.13 on p. 1596). Select the required row in the table and press the **OK** button to close the window. The values in the cells of the selected row will be assigned to the external variables of the file being inserted (or which has already been inserted).

Apart from changing values of external variables of the file inserted into the other document, the table of variables can change the current values of variables of the file containing that table. To assign values from the table of variables to external variables of the file, do the following:.



1. Open the file that contains external variables and a table of variables.
2. Open Variables Window



3. On the Variables Window Toolbar, click the **Table of Variables** button.

The **Table of Variables** window will pop up (see Fig.7.1.13 on p. 1596).

4. Select the required row from the table and press the **Set Values of Variables** button.

The **Table of Variables** window will close and the values from the selected row of the table will be assigned to external variables of the current file.



If the current file is a model, then the **Rebuild** command should be called to rebuild it.



A table will not modify values of the external variables of a file not listed in it.



A table does not modify values of informational variables even if such variables are listed in it.

7.1.5.4.3. Requirements for the Excel file that contains the Table of Variables

Data may be imported from an Excel file into a table of variables. The contents of an Excel file should meet the following requirements to ensure correct importing:

1. The Excel file sheet containing data shall be named *VarTable*. The following system message will be displayed if the sheet is empty or contains invalid data: "Data not Found".
2. There should be no completely empty rows or columns in the table being imported. The data located below the empty row and to the right from the empty column will be ignored at import.

The table of variables is generated in strict correspondence with the Excel table. Rows and columns will be sequenced according to their order in the *VarTable* sheet. The first filled row and column of the *VarTable* sheet will become the first row and column of the table of variables.

3. The cells of the imported table should be filled according to the following rules.
 - ▼ Names of variables are entered into the first row starting from the second cell.
 - ▼ Comments to each set of parameters are entered into the first column starting from the second cell.
 - ▼ Other cells contain variables values. These cells may contain real numbers only.

The message "Wrong format values will be replaced with "0"" will be displayed at import if a table contains invalid data (for example, characters in the value cells). This message also appears if the imported table contains empty or merged cells. Press **OK** button to transfer the modified data or the **Cancel** button to cancel transfer.

Such formatting elements of the imported table as size, style and color of font, size of cells etc., are ignored.

7.2. Adding Constraints to Geometrical Objects

7.2.1. General Information on Parametrization

7.2.1.1. What Is Parametric Image

Unlike conventional image, parametric image contains information not only on location and properties of geometrical objects, but also on relations between objects and on constraints applied to them.

By **interrelation** between objects we mean interdependence between parameters of several objects. Editing one of the interrelated parameters changes the others. If parameters of a certain object are not related to parameters of other objects, then editing of such parameters does not influence any parameters at all. Deleting one or several objects removes their interrelation.

Parallelism and perpendicularity of lines, straight lines, view arrows, offset section line segments, equality of line length or circle radii may be given as examples of relations applied to objects. Slope angles are interrelated parameters of parallel lines, since parallel lines have identical slope angles. Rotating one of such interrelated lines, i.e., changing its slope angle, will rotate the other line as well. If one line is moved or scaled, i.e., its slope angle remains unchanged, then the other line will not change. Deleting one of the lines will make the slope angle of the other line independent.

The relation between parameters may be more complicated than equality of one parameter to the other. For example, it is possible to generate a function determining interrelation of several objects' parameters.

The second type of parametric constraint is **associativity** of objects. Objects may be associative if they are tied to other objects, such as dimensions, technological symbols, and hatches, during development. Such objects “remember” their relation to the reference graphical object (line, circle etc.) or to several objects. Editing reference objects (for example, moving or rotating them) rebuilds the associative objects correspondingly. It results in retention of relative positions of reference and associative objects.

Constraint means relation between an object's parameters or when an object parameter is equal to the constant. Object editing is only allowed if it does not disrupt the existing relations.

Horizontality and verticality of lines, straight lines, view arrows, section view/section lines may be given as an example of constraints applied to geometrical objects. Verticality of a line is identical either to equality of line ends to each other in X-direction, or to its slope angle equality to 90°. With such constraint imposed, the line can be moved but cannot be rotated, i.e., its slope angle can not be changed.

Editing parametrized and associative objects rebuilds the image, observing all constraints applied to the objects and retaining relations between objects.

Parametric images may be used both independently (for example, as a drawing containing parametric views of a part, sketch containing the body section) and for insertion into other documents (drawings or fragments). The parametric image targeted for further insertion must be saved as a fragment (file with *frw* extension) and have **external variables**. More detailed information on external variables can be found in Section 7.1.5.3 on p. 1593; procedure of inserting

parametric fragments is particularized in Section 3.7.2.1 on p. 1248.

7.2.1.2. KOMPAS-3D Parametrization Ideology

There are two conceptually different ways to create a parametric image.

- ▼ Programming or interactive generating of image immediately after drawing. In a number of CAD systems, as an image is generated, drafting rules may be defined simultaneously; however, in case of an error, the defined rule cannot be modified (the entire draft has to be removed and restarted), or its modification may be too complicated.
- ▼ Adding constraints (relations) to objects in the pre-drawn image of an assembly unit or part without following any strict sequence, in arbitrary order. In this case, any model modification not requiring new drafts is possible.

KOMPAS-3D implements the second way of image parametrization. Such parametrization is called variational.

While working on a drawing, fragment or 3D element sketch, various dimensional (linear, angular, radial and diametrical) and geometrical (parallelism, perpendicularity, tangency, attachment of point to a line, point fixing etc.) constraints and relations to image objects may be added.



While working in a drawing, it is possible to create relations between objects (for example, straightening segment vertices) only if these objects belong to the same drawing type. For drawing types, refer to Section 3.5.4 on p. 1160.

7.2.1.3. Parametric features of KOMPAS-3D

This section lists relations and constraints applied to objects manually or automatically; objects created as associative, non-parametrized objects.

In order to make it possible to automatically apply relations and constraints, as well as create associative objects, the parametric mode should be enabled and set up (see Section 7.2.1.8 on p. 1605).

Analytical dependencies between variables are created in the Variables Window (for more information about working with variables, see Section 7.1 on p. 1569).



For convenient use, icons symbolizing constraints applied to objects and icons showing the number of degrees of freedom these objects have, may be enabled (see Section 7.2.5 on p. 1620).

7.2.1.3.1. Relations and constraints

KOMPAS-3D gives the user an opportunity to add the following relations and constraints:

- ▼ Verticality of objects*
- ▼ Horizontality of objects*
- ▼ Collinear lines*
- ▼ Parallelism of objects*

- ▼ Perpendicularity of objects*
- ▼ Aligning object handles vertically*
- ▼ Aligning object handles horizontally*
- ▼ Mirror symmetry*
- ▼ Placing objects on an angle bisector
- ▼ Equal arc and circle radii
- ▼ Equal length of lines
- ▼ Line tangency*
- ▼ Merging object handles*
- ▼ Point on line*
- ▼ Placing of Line Midpoint
- ▼ Fixing object handles
- ▼ Length fixing
- ▼ Angle fixing
- ▼ Fixation of a dimension* and editing its value.
- ▼ Assigning a variable name to a dimension*

Relations and constraints marked with an asterisk can be applied automatically when objects are created in certain ways or with the use of snaps, as well as when performing the following constructions:

- ▼ Fillet
- ▼ Chamfer
- ▼ Mate
- ▼ Trim
- ▼ Trimming by two points
- ▼ Setting points on an arc
- ▼ Symmetry

Thus, for example, as a result of cutting a line segment by another one, the trimmed segment will be linked to the trimming segment by means of the *point on curve* relation.

7.2.1.3.2. Associative objects

You can enter associative geometrical objects and designations. These include:

- ▼ hatches and fills;
- ▼ Surface finish symbols
- ▼ Datum symbols
- ▼ marks/reference designations on the line
- ▼ Dimensions
- ▼ Center marks
- ▼ Offset curves

▼ Conditional intersections.



All associative objects, except for linear dimensions, have the *associativeness* constraint. This constraint is applied to objects automatically during their creation; it is impossible to apply it manually.

At the same time, linear dimension is considered to be associative if starting points of its extension lines coincide with points of a geometrical object. Therefore, non-associative linear dimension can be manually associated with the required object by applying the *coincidence* constraints to the dimension's extension lines.

7.2.1.3.3. Non-parametrized objects

Constraints cannot be added to the following composite objects:

- ▼ rectangle,
- ▼ polygon,
- ▼ polyline,
- ▼ Bezier curve,
- ▼ contour,
- ▼ text.
- ▼ table,
- ▼ extension line any kind,
- ▼ macro element,
- ▼ drawing's layout.

It is impossible to apply relations and constraints to points and lines of non-parameterized objects; dimensions assigned to them cannot be associative.

7.2.1.4. Adding Relations and Constraints Principles and Techniques

Several constraints and relations may be specified without apparent input of numerical values (for example, horizontality of a line or tangency condition for two lines). Conversely, such constraints as dimensions shall have numerical values. Some relations and constraints may be defined as an equation (for example, a function of an object parameter dependence on parameters of other objects may be defined).

Some constraints and relations (coincidence of points, parallelism etc.) may be generated automatically at input if the user has enabled this option. For example, points merge or locating a point on a straight line are parametrized through a snap that was set upon creating of a point (global or local, see Section 3.1.1 on p. 865), and conditions of parallelism, perpendicularity and tangency— through corresponding processes of object input. Automatic generation of constraints and relations may be disabled at any time.

Additional mutual relations and constraints can be assigned to drawing object at any time while working with the document.

Associativity of objects (dimensions, hatches etc.) appears only at input due to direct or indirect indication of reference object. There are no special commands for defining associativity.

Constraints are added by selecting their type and specifying an object for parametrization.

For example, you select horizontality and specify the lines that should be horizontal.

The relations are added by selecting their type and specifying a pair of interrelated objects.

For example, you choose parallelism and specify the line pairs that should be parallel.

At that, only the relations that were added apparently (either by calling the corresponding parametrization command or by drawing in parametric mode) will be memorized. Note that new relations resulting from several previously applied ones will not appear automatically even if they seem to be quite evident.

For example, you have drawn three lines and specified parallelism between the first and the second line as well as between the second and the third line. At that, the relation between the first and the third line is mediated, for these lines are related through the second one. Upon deleting the second line, the first and the third ones will be parallel. However, such lines will be independent from each other at editing, as there is no direct relation between them.

Another method of adding constraints is fixing an associative dimension. If a dimension is associative, it can be fixed using the corresponding command. The value of a fixed dimension stays constant for any rebuilding of objects that form an image. Dimension value always characterizes some geometrical parameter. For example, the value of an associative linear dimension corresponds to the distance between the handles of objects (or a single object), and the value of associative radial dimension—to the radius of a reference circle or an arc. Thus, fixing a dimension allows to set some object parameter equal to constant. A value of every certain fixed dimension can be modified using a special command, but it cannot be modified by editing objects themselves.

If a relation between parameters of objects should be specified in analytical form, the variables corresponding to dependent parameters should be created first. Subsequently, equations may be introduced using created variables.

To create a variable, it is necessary to set an associative dimension that characterizes the parameter being limited (for example, it will be a linear dimension between line ends for the line length), and to assign a variable name to this dimension using the corresponding command.

Upon assigning a variable name to the fixed dimension, the value of this dimension may be used in expressions.



Non-fixed associative dimension is information only. Unlike fixed dimensions, an information dimension value depends on dimension and location of geometric objects. Assign a variable name to the info dimension to use it in calculations.

More about fixed and information dimensions, see Section 7.2.1.10 on p. 1606.

The expressions are to be entered in the Variables Window.

Constraints and relations removing procedure is explained in Section 7.2.3 on p. 1617.

7.2.1.5. Tips on Using Parametrization Capabilities

Parametrization capabilities do not apply any restrictions on working style when creating drawings. You may choose the image you prefer, parametrized or conventional. If necessary, parametric and nonparametric objects can be combined in one document. Additionally, you can

easily switch between geometry representations, for example, by adding constraints to the existing conventional drawing or by removing constraints from the existing parametric image.

When working with drafted documentation, use parametric capabilities discreetly, evaluating real benefits of adding a full set of constraints to a particular drawing.

There are some general recommendations related to drawings parameterization.

- ▼ It is beneficial to make parametric drawings of parts whose modifications include only dimension changes, not changes in the image topology. Thus, once created, a parametric image of a part may be quickly rebuilt simply by changing dimension values.
- ▼ When starting a new design, decide whether it will serve as a template in the future. If it will not, no drawing parametrization is needed since no future quick modifications are required. If, on the contrary, the new part will be often used as a standard template, making its drawing parametrization or creating a parametric fragment is beneficial.
- ▼ Full parametrization of complex assembly drawings will probably be not justified as it demands too much effort on adding constraints and controlling dimensions.
- ▼ Try to estimate benefits that may be achieved by parametrization using specific samples of drawings typical for your enterprise. In the future, take this into account when developing a new design or when revising existing drafting documentation.

7.2.1.6. Specifics of Working with Parametric Objects

When working with parametric drawings and fragments, it may be useful to know the following specifics.

- ▼ The more constraints are applied to model objects, the less probable is a large variability in a series of calculations. Usually, it is recommended to define the image entirely, i.e., to remove all degrees of freedom from its constituent objects. As additional constraints, you may use point fixing, applying horizontal or vertical condition on line segments, or creating additional dimensions. Degrees of freedom can be displayed for your convenience (see Section 7.2.5.2 on p. 1624).
- ▼ It is recommended to refrain from “big moves” when editing a parametric image; “step-by-step” is the best approach to such work. For example, dimension value change shall not be excessive (i.e., from 5 to 120 degrees). It is better to implement such considerable changes sequentially, in several steps. This also relates to editing by dragging points— do not move an object or point too far away at once, such a move should be executed in a series of steps.
- ▼ Remember that parametric model processing time significantly depends on the drawing’s or fragment’s degree of population with parameterized objects. However, a fully defined model is processed faster than partially defined.

7.2.1.7. Parametric Mode

The parametric mode is a mode of creating and editing geometrical and layout objects in which parametric relations and constraints are applied automatically. The type of applied relations and constraints is defined during drafting based on the object drawing command sequence or on engaged snaps (including local ones).

Some examples of working in parametric mode are shown below.

- ▼ Drawing a line segment parallel to another line by the **Parallel Line** command while in the parametric mode will cause automatic creation of the respective relation, that of the *parallelism* of lines.
- ▼ If, when drawing a circle in the parametric mode, its center will snap to an end of line, then the respective relation— *merged points*— will be automatically created.
- ▼ Drawing a vertical line segment in the parametric mode results in the respective constraint *verticality*, that is applied to the line.
- ▼ Creating a horizontal cut line with circle center alignment adds the constraint *horizontality* to a cut line and generates the relationship *points horizontal alignment* between the first line dash and circle center.
- ▼ Entering the surface finish symbol results in *associative* surface finish symbol creation.
The parametric mode is disabled for fragments and drawings by default; it is enabled for 3D element sketches.



To enable parametric mode is recommended when making drawings that contain associated views. It will allow creating associative design objects (dimensions, center marks, surface finish symbols etc.) which will “follow” their graphic objects when rebuilding the latter after editing the model.

7.2.1.8. Enabling and Setting Up Parametrization Mode



To enable parametrization mode in the current graphic document, click **Parametrization Mode on the Current Status panel**. This done, the associativity of all objects, build-ups parametrization, and dimensions fixation will be initiated by default.

In order to change default settings of the parametrization mode, use the dialog which is called by the following command: **Service — Parameters... — Current drawing/fragment — Parametrization** (see Section 9.2.6.21 on p. 1830).



Activation of parametrization mode will have no effect, if you keep disabled all options in the dialog window for groups **Associate at Input:** and **Parametrize:**. In this case, no relations and restrictions will be applied, i.e. patterning will be carried out in the mode as if the parametrization mode had been disabled.

The settings of the parametric mode are saved in the document itself and are not modified when transferring the document to the other work place.

Customizing every individual graphical document if most of them are developed in the parametric mode is counterproductive. In this case, parametric mode may be set as default for all graphical documents.

To do this, call the **Service — Parameters... — New Documents — Graphic Document — Parametrization**, and make appropriate settings.

7.2.1.9. General Algorithm for Creating Parametric Images

With parametric mode is enabled (see Section 7.2.1.7 on p. 1604), the relations and constraints that were selected in the parametrization properties dialog are automatically generated by system at object input.

No action is required when inputting objects. However, remember that points object coincidence is parameterized via snaps that were activated when selecting those points. It does not matter which snap was active— a global or a local one. A point selected simply “nearby” of another point without activating a snap, will not be parameterized. Coincident points are also introduced when dragging object handles (also using an active snap).



“Point” in this sense is not understood as a geometrical object, but rather as any handle of an object introduced at its creation (a line segment start and end point, a circle or ellipse center, etc.)

Associative design objects are defined according to the normal procedure.

Various additional mutual relations and constraints may be assigned to drawing objects at any time if necessary.

Analytic dependences between variables (related with dimensions) are defined in the Variables Window. If dimensions belong to the current drawing type, this type will be automatically reconstructed after you change variable values in the Variables Window. To rebuild non-current types, call the **View — Rebuild** command or press <F5> after you have finished editing variables or expressions.

7.2.1.10. Controlling and Informational Dimensions. Dimensions with Variables

Two types of associative dimensions are available: controlling and informational. The dimension's type defines whether the dimension will affect the geometrical object or, vice versa, the object will affect the dimension.

Controlling dimension is the dimension which controls geometrical object.

The value of the controlling dimension can be modified by the user by entering a number, constant, or expression (in the two latter cases, a variable should be created for the dimension). After that geometrical object (to which the dimension was set) is rebuilt to satisfy this new dimension value.

To make a dimension the controlling one, it has to be fixed, i.e. have the *fixed dimension* constraint. Rectangular frame around its value indicates that the dimension has been fixed.



Automatic fixing of dimensions at input is available in the parametric mode. For that, at parametric mode setup (see Section 7.2.1.8 on p. 1605) the options **Associate at Entry: Dimensions** and **Fix Dimensions** should be enabled.

Besides that, you can fix dimensions using the **Fix dimension** command (see Section 7.2.2.17 on p. 1613).

The object (for which fixed dimension was set) can be edited only in such a way as to satisfy this constant value of the dimension. For example, a line can only be moved and rotated (its length cannot be changed) if the dimension fixing its length was set for this line.

Info dimension is the dimension which is controlled by geometrical object.

A value or an expression cannot be entered in info dimension. Geometrical object (for which info dimension is set) can be edited as you wish. After geometrical object has been rebuilt its dimension is redrawn and its value is recalculated. The informational dimension has no *fixed dimension* constraint.



When editing geometrical objects not only their dimension are taken into account but also their parametric relations and constraints

Any controlling dimension can be converted into informational by two methods:



- ▼ Delete the *fixed dimension* constraint using the command **Show/Delete Constraints** (see Section 7.2.3.1 on p. 1618),
- ▼ Enable the option **Info Dimension** in the dimension setup dialog (see Section 7.2.2.20 on p. 1614).



Info dimension can be made the controlling one using either of the two methods:

- ▼ Use the **Fix Dimension** command (see Section 7.2.2.17 on p. 1613),
 - ▼ Disable the option **Info Dimension** in the dimension setup dialog window.
- However, not every informational dimension can be made the controlling one, i.e. become fixed. The following reasons may cause the fact that dimension can not fixed:
- ▼ Geometrical relations between objects, for example: The distance between diagonally opposite vertices of a rectangle depends on the lengths of its sides, that is why the length of its diagonal cannot be fixed if two of its sides have already been fixed.
 - ▼ Parametrical relations and constraints, for example:
 - ▼ Angle between two lines to which the *Perpendicular* relation has been applied is always equal to 90 degrees, that is why this angle cannot be fixed.
 - ▼ The diameter of the circle, mating parallel lines, always equals to the distance between them, therefore, if the distance is already fixed, it is impossible to fix the diameter of the circle.

Redundant dimensions cannot be fixed.

The corresponding message will pop up if you are about to fix the dimension which can be info only.

You can create a variable both for the controlling and the informational dimension. Enter a variable name in the dimension setting dialog (see Section 7.2.2.20 on p. 1614).

The dimension for which the variable is created has the *dimension with variable* constraint. Deleting the constraint results in removal of the variable.

7.2.2. Adding Relations and Constraints

Capabilities, the command button, procedure of indicating objects for parametrizing as well as principles of operation in the absence of other relations and constraints are given in the description of every command. If a relation or constraint added by command may appear automatically in parametric mode, then the description indicates the correspondent option of parametric mode setup.

With relations and constraints already existing in the objects involved into parametrization, new relations and constraints will be added considering the existing ones. At that, the image is rebuilt observing all relations and constraints with minimum object parameters modifications. Relations and constraints that conflict with the existing ones will not be added.

Commands for adding the above-listed relations and constraints are grouped in the menu **Tools — Adding constraints**, and buttons for calling the commands — on the toolbar **Adding constraints** (Fig. 7.2.1).

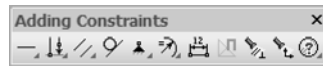


Fig. 7.2.1. **Adding Constraints** toolbar



Note that some objects cannot be parametrized (see Section 7.2.1.3.3 on p. 1602).

You may enable and customize display of symbols that indicate relations and constraints applied to graphical objects, if necessary (see Section 7.2.5 on p. 1620).

7.2.2.1. Horizontality



To transform inclined objects (lines, view arrows etc.) into horizontal, call the command **Horizontality**.

Show the sequence of objects requiring transformation.

When a line is transformed into horizontal, its X-component remains constant, i.e., a line is rotated without changing X-coordinates of its ends. As a result, inclined line length decreases, and horizontal line length does not change. A vertical line can not be transformed into horizontal, as this leads to its transformation into a line of zero length.



An object parallel to the abscissa axis of the coordinate system of the current view is considered to be horizontal. Therefore, if you add *horizontality* constraint to an object that is located in a view with coordinate system rotated relatively to the absolute coordinate system, then such an object will not be parallel to the horizontal sides of the sheet (for the information about views of drawing, see Section 3.5.4 on p. 1160).

The given constraint— *horizontality*— may be automatically added to an object being created in parametric mode. For that, in parametric mode setup (see 7.2.1.7 on p. 1604) the **Add Constraints: Horizontal and Vertical** option should be enabled. The constraint will be added to

a horizontal object while creating by any method. For example, when specifying the second point of a line (or a straight line), the snap **Alignment** may be assigned to the first point, or the fields of the Property Bar to enter the identical Y-coordinates of handles may be used, or a zero slope angle may be entered. The *horizontality* constraint will be added to any horizontal object created, even if its horizontality was not set explicitly (for example, a new line was drawn parallel to the horizontal object).

7.2.2.2. Verticality



To transform inclined objects (lines, view arrows, etc.) into vertical, call the **Verticality** command.

Show the sequence of objects requiring transformation.

When a line is transformed into vertical, its Y-component remains constant, i.e., a line is rotated without changing Y-coordinates of its ends. As a result, inclined line length decreases, and vertical line length does not change. A horizontal line can not be transformed into vertical, as this leads to its transformation into a line of zero length.



An object parallel to the ordinate axis of the current view coordinate system is considered to be vertical. Therefore, if you add *vertical* constraint to an object that is located in a view with coordinate system rotated relatively to the absolute coordinate system, then such an object will not be parallel to the vertical sides of the sheet (for the information about views of drawing, see Section 3.5.4 on p. 1160).

The given constraint— *verticality*— may be automatically added to an object being created in parametric mode. For that, at parametric mode setup (see Section 7.2.1.7 on p. 1604) the **Add Constraints: Horizontal and Vertical** option should be enabled. The constraint will be added to a vertical object being created by any method.

7.2.2.3. Horizontal Alignment of Points



To align handles of geometrical objects horizontally, call the command **Align Horizontally**.

Select pairs of handles of geometrical objects to merge.

To execute

- ▼ **Align Handles Horizontally,**
- ▼ **Align Handles Vertically,**
- ▼ **Merge Points,**
- ▼ **Line Midpoint**
- ▼ **Point on Line,**
- ▼ **Symmetry of Two Points**

only those points may be indicated which belong to the current view (see Section 3.5.5.2 on p. 1171 for the information on the view status change).

7.2.2.3.1. Selecting the Point for Alignment from Several Coincident Points

Sometimes the point that should be indicated for alignment coincides with a handle of another object. It is impossible to choose such a point accurately, as snapping to the point of another object hampers it. In this case, you may first indicate the graphical object which the point being parametrized belongs to, and then a point itself.



To do this, click the **Select Object** button.

Indicate the object and its handles.

Selecting one of the coincident points is performed identically when executing the commands **Align Handles Vertically** and **Merge Points**.

7.2.2.4. Vertical Alignment of Points



To align handles of graphical objects vertically, call the **Align Handles Vertically** command. Select pairs of handles of geometrical objects to merge.

Relations *align handles horizontally* and *align handles vertically* may be automatically added to objects being created in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7 on p. 1604) the **Snapping** and **Alignment snapping** options should be enabled. The relations will appear at object handle input by using snapping **Alignment** to the other handle.

7.2.2.5. Merging Points



To snap handles of geometrical objects to each other, call the **Merge Points** command.

Select pairs of object handles to merge.

The given relation— *merge points*— may be automatically added to objects being created in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7) the **Snapping** option should be enabled. The relation will appear at object handle input by using snapping to another handle, and also by calling the command **Continuous input** to draw a sequence of objects (see Section 3.2.10 on p. 928).

7.2.2.6. Point on Curve



To snap an object handle to another curve (to another object), call the **Point on Curve** command.

Indicate first a curve on which the point should be placed, and then the point.

The given relation— *point on curve*— may be automatically added to objects being created in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7 on p. 1604) the **Snapping** option should be enabled. The relation will appear at object handle input by calling the snap **Point on Curve**, when drawing a circle using the **Circle Centerpoint on Object** command (see 3.2.5.3 on p. 911 Section), as well as when drawing points on a curve using the **Points on Curve** (see Section 3.2.2.2 on p. 899) and **Points at Specified Distance** (see Section 3.2.2.5 on p. 900) commands.

7.2.2.7. Point on Curve



To snap an object handle to another curve (to another object), call the **Line Midpoint** command.

Indicate first a curve (a segment or a circular arc) on which the point should be placed, and then the point.

The given relation—*point in the middle of a curve*—may be automatically added to objects being created in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7 on p. 1604) the **Snapping** option should be enabled. The relation will appear at object handle input by calling the snap **Midpoint**.

7.2.2.8. Symmetry of Two Points



To set symmetry of object handles with respect to the axis (mirror symmetry), call the **Symmetry of Points** command.

Select an axis of symmetry, and then a pair of object handles that need to be symmetrical.

The given relation—*mirror symmetry*— may be automatically added to objects being created in parametric mode. For that, during parametric mode setup the **Symmetry** option should be enabled. The relation will appear at calling the **Symmetry** command (see section 3.4.4.3 on p. 1131), providing that an existing line (rather than two points belonging to a symmetry axis) is specified as a symmetry axis.

7.2.2.9. Parallel



To set objects (lines, view arrows etc.) parallel, call the **Parallel** command.

The objects that are to be set parallel should be indicated by pairs.

To execute

- ▼ **Parallel,**
- ▼ **Perpendicular,**
- ▼ **Collinearity,**
- ▼ **Bisector,**
- ▼ **Tangency,**
- ▼ **Equal Radius,**
- ▼ **Length equality.**

only points belonging to the same view may be specified (for the information on working with views of drawing, see Section 3.5.5 on p. 1170).

The given relation—*parallelism*— may be automatically added to objects being created in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7 on p. 1604) the **Parallelism** option should be enabled. The relation will appear at calling **Parallel Straight Line** (see section 3.2.3.4 on p. 903) and **Parallel Line** (see section 3.2.4.2 on p. 906) commands to create lines and straight lines.

7.2.2.10. Perpendicular



To set objects (lines, view arrows etc.) perpendicular, call the **Perpendicular** command.

The objects that are to be set perpendicular should be indicated by pairs.

The given relation— *perpendicularity*— may be automatically added to objects being created in parametric mode. For that, at parametric mode setup (see Section 7.2.1.7) the **Perpendicularity** option should be enabled. The relation will appear at calling **Perpendicular Straight Line** (see section 3.2.3.5 on p. 903) and **Perpendicular Line** (see section 3.2.4.3 on p. 907) commands to create lines and straight lines.

7.2.2.11. Collinear



To set collinearity of lines, call the command **Collinear**.

The objects that should lie along a straight line should be indicated by pairs.

The given relation— *collinearity*— may be added automatically when creating objects in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7 on p. 1604) the **Snapping** option should be enabled. The relation will appear at calling **Trim Curve by Two Points** (see section 3.4.7.2 on p. 1138) command to divide a line into two lines.

7.2.2.12. Bisector



To position the object (line, view arrow etc.) on a bisector of an angle formed by two selected rectilinear objects, click **Bisector**.

Indicate three objects at once in the following order.



At first, specify two objects making up an angle, to whose bisector there should be a link. The angle bisector ghost will appear. Bisector options are displayed one-by-one when clicking **Next Object** and **Previous Object** buttons of the Special Control Panel.



After the required bisector option appears on the screen, specify the object being placed.



If two objects are parallel to each other, the third object is located in the middle between them.

The given relation— *bisector*— may be added automatically when creating objects in parametric mode. For that, during parametric mode setup (see Section 9.2.6.21 on p. 1830) the **Bisector** option should be enabled. The relation will appear when constructing a straight line using **Bisector** command (see Section 3.2.3.9 on p. 905), an auto centerline by two line segments using **Auto Centerline** command (see Section 3.3.7.16 on p. 1050), a line segment on an angle bisector using **Line** command (see Section 3.2.11.2.6 on p. 935).

7.2.2.13. Tangency



To set tangency of curves, call the **Tangency** command.

Select the first and the second line whose tangency needs to be set.

The given relation— *tangency*— may be added automatically when creating objects in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7) the **Tangency** op-

tion should be enabled. The constraint will be added to a tangent object being created by any method.



When applying the *tangency* relation to NURBS, note the following peculiarities:

- ▼ tangent point of NURBS with another curve is the point in which NURBS was specified,
- ▼ if you need the point of tangency to be the outermost point of NURBS, position the point on the curve which NURBS should be tangent to prior to applying the relation (applying the *point on curve* relation is not obligatory).

7.2.2.14. Equal Radius



To make radii of the indicated arcs and/or circles equal, call the **Equal Radius** command. Select pairs of arcs and/or circles to set equal radii.

7.2.2.15. Equal Length



To make lengths of the indicated lines equal, call the **Equal Length** command. Select pairs of line segments to set equal lengths.

7.2.2.16. Point Fixing



To fix the coordinates of geometric objects' handles, call the **Fix Point** command. Select the geometric object handles to be fixed.

This relation—*fixed point*—may be added automatically when creating objects in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7) the **Snapping** option should be enabled. The relation will appear at inputting an object handle that coincides with the coordinate origin using **Nearest point** snapping.

7.2.2.17. Dimension Fixing



To fix the value of the indicated dimension, call the **Fix Dimension** command.

Indicate the dimensions to be fixed. The dimension text is enclosed into rectangular frame, which is displayed if display of constraints is enabled in the current window (see Section 7.2.5 on p. 1620) upon successive fixing. This frame is not printed out.

Fixing is available for dimensions of all types, excluding the following:

- ▼ Linear dimension with a broken line
- ▼ Arc dimension
- ▼ Height dimension for the top view with leader
- ▼ Height dimension on the drawing itself.

It is impossible to fix non-associative or excessive dimensions. The excessive dimension is such a dimension whose value can be calculated either from equations existing in the document, or from controlling dimensions that were previously set.

Automatic fixing of dimensions at input is available in the parametric mode. For that, at parametric mode setup (see Section 7.2.1.7 on p. 1604) the options **Associate at Entry: Dimensions** and **Fix Dimensions** should be enabled. The relation will appear at associative non-successive dimensions input.



During automatic fixation names of variables are also assigned to the dimensions.

7.2.2.18. Length fixing



To fix the length of the line or the centerline, call command **Fix the Length**.

Specify the objects to fix their length.

Automatic fixing of length at input is available in the parametric mode. For that, at parametric mode setup (see Section 9.2.6.21 on p. 1830) the **Add Constraints: Tangency** option should be enabled.

7.2.2.19. Angle fixing



To fix inclination angle of an object (line segment, centerline, break line etc.), call command **Fix an Angle** In a fragment, inclination angle of object is fixed in the absolute coordinate system, in a drawing—in the coordinate system of view in which object is placed.

Select objects for fixing their inclination angles.

This relation—*fixed angle*—may be added automatically when creating objects in parametric mode. For that, during parametric mode setup (see Section 7.2.1.7) the **Snapping** and **Angle snapping** options should be enabled. The relation will appear at object handle input by calling the **Angle snapping** snap.

7.2.2.20. Dimension Value Setting

Enter a controlling dimension value in the dimension value setting dialog (Figure 7.2.2). This dialog also makes it possible to create a variable related to dimension. This variable is used for controlling the dimension's value.

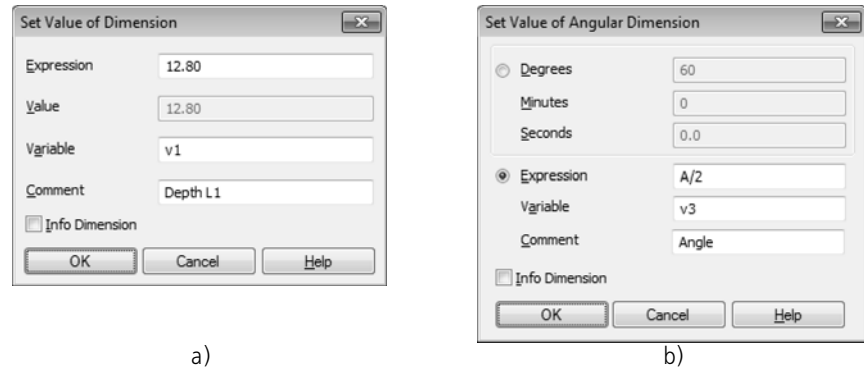


Fig. 7.2.2. Dialog to set dimension value:
a) linear, diametric or radial; b) angular

The dialog for setting the dimension's value appears automatically when assigning the dimension. Besides that, the dialog can be called as follows:



- ▼ by the **Set dimension's value** command,
- ▼ by double-clicking the dimension's dimension text.



Note that double-clicking any other part of a dimension (for example, on a dimension line) initiates the dimension editing process.

In order to set the value of dimension, do the following.

In order to set the value of a linear, diameter or radial dimension, specify the method of getting the value in the **Expression** field: a number, constant, or expression for calculation of the value (the expressions' syntax is described in Section 7.1.3.3.1 on p. 1580),

The angle dimension's value can be set using either of the two methods.

In order to set the value in the $XX^{\circ}XX'XX''$ format, enable the **Degrees** option and enter numeric values in the **Degrees**, **Minutes**, and **Seconds** fields. Note that none of the fields should be left empty. If you need a zero value, enter zero in this field.

In order to set the value in the XX,XXX° format, use the **Expression** field by enabling its respective option. Value setting options for this field are described above.

The prescribed value of the dimension is also the value of the variable related to that dimension.

In order to create a variable for the dimension, enter its name in the **Variable** field. The name of variable created by the template "vN" is offered in the dialog by default, where N— the number of the variable in the list of model's variables. If necessary, you can edit the name of the variable. It may contain letters of the Latin alphabet (case-sensitive), western numerals and the underscore characters ("_"). The length of a variable name may not exceed 512 characters. The first character in the variable name is a letter or an underscore.

You can use the **Commentary** field to specify a commentary for a variable.

If you need to create a dimension without a variable, delete the variable's name specified in the **Variable** field. In this case, the variable will not be created.

Option **Info Dimension** allows selecting either information or controlling dimension type. If the option is enabled, the informational dimension type is used. It is impossible to manually set value for this type of dimension. If this option is disabled, the dimension is controlling. The value of this type of dimension is set as described above. For more about controlling and information dimensions, see Section 7.2.1.10 on p. 1606.



When creating a dimension without a variable or an informational dimension, note the following peculiarity.

If the dimension's value is set by a constant or by an expression, the constant's/expression's designation will not be saved after closing the dialog by clicking **OK**. The **Expression** field contents is replaced by the calculated value of the dimension which is displayed in the field when the dialog is reopened.

For a dimension without a variable, commentary is also not saved (the **Commentary** field becomes empty).

Having performed all the desired operations in the dialog, close it by clicking **OK**. All changes will be saved.

The value prescribed for the dimension is transferred to the document while all information about the created variable is transferred to the Variable's window (see Section 7.1.1.3 on p. 1571).

The variable is displayed in the dimension's dimension text when the constraints display mode is enabled (see Section 7.2.5.1 on p. 1620). Peculiarities of displaying dimensions with variables are described in Section 7.2.5.1.1 on p. 1623.



If you finish working with the dialog by clicking **OK**, the following problems are possible.

- ▼ it is impossible to fix or apply a new value to the dimension; for example, due to applied constraints,
- ▼ the dimension can only be the informational one, but the **Informational dimension** option is not enabled in the dialog.

If one of the above-mentioned problems arise, the respective message appears on the screen. The dialog for setting the dimension will remain open.

7.2.2.21. Object Parametrization



To add some types of relations and constraints to geometrical objects semi-automatically, select them (see Section 3.1.3 on p. 876 for object selection methods) and call the **Add constraints** command.

Parametrization setup dialog will pop up (Fig.7.2.3). The control elements of this dialog are represented in Table 7.2.1.



Automatic object parametrization in associated views, maintaining relation with the models, is impossible.

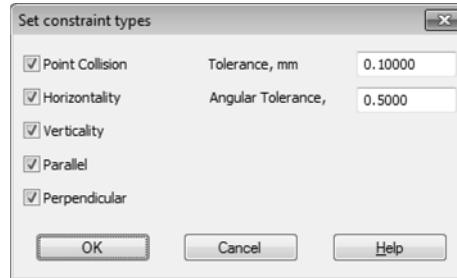


Fig. 7.2.3. Selected object parametrization setup dialog

Tab. 7.2.1. Control elements of the parametrization setup dialog.

Element	Description
Merge Points Horizontality Verticality Parallel Perpendicular	Enable the options that correspond to the types of relations and constraints that are to be added to the selected objects.
Tolerance, mm	A field for points merging tolerance input. If the distance between two handles of objects being parametrized is less than the specified tolerance, then such handles will be merged*.
Angular Tolerance, deg	Field for angular tolerance input. If a horizontal or vertical deviation angle of lines or straight lines being parametrized is less than angular tolerance, and if the options Horizontal and Vertical are enabled, then such objects will become horizontal and vertical respectively*. If deviation of parametrized lines or straight lines from parallelism or perpendicularity is less than the specified tolerance, and if the options Parallel and Perpendicular are enabled, then such objects will become parallel or perpendicular respectively*.

* The corresponding command for adding the constraints and relations to objects will be automatically initiated.



If many objects are selected, the command execution time may be considerably long.

7.2.3. Viewing and Deleting Relations and Constraints

Prior to initiating commands of viewing and deleting relations and constraints, it is necessary to select the object whose relations and constraints are to be viewed or deleted. For more de-

tails about selecting objects, see Section 3.1.3 on p. 876.

If the selected object has no relations and constraints, then commands for viewing and deleting them will not be available.

Several objects may be selected before calling the command for deleting all constraints.



You may enable display of constraints applied to graphical objects (see Section 7.2.5 on p. 1620). Special symbols indicating objects constraints are displayed in this mode.

7.2.3.1. Show/Remove Constraints



To view the list of relations and constraints applied to the selected object and delete any of them, call the **Show/Remove Constraints** command.

A list of relations and constraints of the selected object will appear on the Property Toolbar.

Selecting any relation in the list by clicking on it will highlight the related object. This is the way to control the choice of the required relation from the list of similar ones.



To delete the selected relation or constraint, click the **Delete** button.

To view relations and constraints applied to another parametrized object, use the mouse to select it in a document window.

7.2.3.2. Delete All Constraints



To delete all relations and constraints applied to the selected object(s), call the **Delete all constraints** command.

As a result, the selected objects will lose all relations and constraints.

7.2.4. Techniques for Working with Parametric Images

7.2.4.1. Converting a Conventional Image into Parametric

To make dimensions of a hatches, surface finish symbols and datum symbols associated with geometrical objects, it is possible to reselect base curves and points in the respective editing modes. Another method is to delete old non-parametric objects and create them anew (highly recommended for hatches in areas of complicated configuration). In both cases, the parametric mode should be enabled in advance (see Section 7.2.1.7 on p. 1604).

Some of parametric constraints for objects (merge points, horizontality, verticality, parallelism and perpendicularity) can be added semi-automatically. For that, use the **Add Constraints** command (see Section 7.2.2.21 on p. 1616).

Other types of relations and constraints should be added to non-parametrized objects manually.

7.2.4.2. Converting a Parametric Image into Conventional

To fully convert a parametric drawing or a fragment into a conventional one, do the following.



1. Select all objects.
2. Call the **Delete All Constraints** command.

It is recommended to save, close and then reopen the document after removing all constraints from objects.

7.2.4.3. Parametric Image Editing

7.2.4.3.1. Variables Values Modifying

Values of variables can be changed in the Variables Window (see Section 7.1.1.2 on p. 1569). A corresponding message will be displayed if a set of equations becomes insoluble with the variable specified.

Values of variable dimensions of graphical documents can be preset in the dimension setting dialog (see Section 7.2.2.20 on p. 1614).

7.2.4.3.2. Dimension Values Controlling

Modification of dimension value (and, thereby, of configuration of the geometry that dimension value controls) is available only if that dimension is fixed.

To modify the value of dimension, do the following:

1. Double-click the dimension text of the parametric dimension. The dialog to set the dimension value will be displayed.
2. The field **Value** is available if the dimension is fixed. Enter a new dimension value.



If a variable name is assigned to the dimension, you can enter an expression to calculate the dimension value, or, visa versa, replace the existing expression by a numeric value (however, the variables being a part of the deleted expression, will not be deleted from the document).

Disable the **Info Dimension** option if the dimension is not fixed. It will fix the dimension and enable the **Value** field. Enter a new dimension value.

3. Click the **OK** button. The dimension value will change.

A respective message will be displayed if fixing a dimension or changing its value is impossible due to excessive constraints.

7.2.4.3.3. Editing by Dragging Points

You can edit a parametric image with the mouse by “dragging” handles of component objects (see Section 3.4.1.1.3 on p. 1114).

Moving the object handle will also rebuild all related objects.

If constraints applied to an object fully define its position (for example, all necessary dimensions that drive the part geometry are applied), then the related objects will be simply moved into the new position without rebuilding the geometry. If, additionally, the object is related to a fixed point, then no actions at all will be performed.

Enabling degrees of freedom display makes it easier to determine whether handles can be dragged with a mouse (see Section 7.2.5.2 on p. 1624). If there is at least one degree of freedom present, then the handle can be dragged; within absence of all degrees of freedom, dragging is impossible.



Degrees of freedom of an object are displayed even when the image is completely determined, and moving a handle will simply move the image.

7.2.5. Display of Constraints and Degrees of Freedom

It is much more convenient to work with parametric image (for example, a sketch), if, along with the geometrical objects, their applied constraints and degrees of freedom are also displayed.



The following commands serve to display constraints and degrees of freedom:

▼ **Display Constraints,**



▼ **Display Degrees of Freedom.**

Constraints and degrees of freedom are displayed on the screen as long as the respective buttons are pressed. To disable all symbols of a certain type, release the respective button or call the command again. You cannot disable constraints or degrees of freedom for an individual object.



In drawings, constraints and degrees of freedom are displayed only for the objects that belong to the current view.

Note that constraints and degrees of freedom display setting affects only the current window. You can enable or disable the symbols displayed in other windows (including another window of the current document) independently from the current window.



In drawings and fragments degrees of freedom are shown only for the objects that have constraints, whereas in sketches degrees of freedom are shown for all objects.



Display of constraints in drawings and fragments is disabled by default and is enabled in 3D sketches.

You may use the constraints and degrees of freedom setup dialog to specify color of symbols and enable (disable) constant display of variables, fixed dimensions frames and fixed points symbols. Working with the dialog is described in Section 9.1.7.7 on p. 1709.








7.2.5.1. Constraints



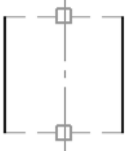




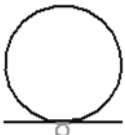

To display the symbols of relations and constraints applied to graphical objects, call the **Display Constraints** command.

Upon calling the command, the constraint and relation symbols will appear (see Table). 7.2.2).






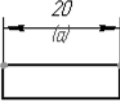
Tab. 7.2.2. Graphical objects relations and constraints symbols

Constraint	Symbol	Position
Horizontality		Line midpoint
Verticality		Line midpoint
Handles alignment Horizontal		Dashed horizontal segment connecting aligned points
Handles alignment Vertically		Dashed vertical segment connecting aligned points
Merge Points		At the point
Point on Line		At the point; if necessary, the curve is extended with a dotted line
Curve Midpoint		At the point

Tab. 7.2.2. Graphical objects relations and constraints symbols

Constraint	Symbol	Position
Symmetry of Two Points		Dashed line connecting symmetrical points and a square in the point of intersection of that line with an axis; an axis is extended to the center of the square with a dashed line if necessary
Parallel		Line midpoints
Perpendicular		At Points intersection; the line segments are extended with a dashed line if necessary
collinearity		Above mid-segments; the nearest ends of line segments are connected with a dashed line
Bisector		Above the midpoint of an object
Tangency		At tangency point; the line segments are extrapolated up to the tangency point with a dashed line if necessary
Equal Radius		For an arc— above the midpoint, for a circle— above the top point

Tab. 7.2.2. Graphical objects relations and constraints symbols

Constraint	Symbol	Position
Equal Length		Above mid-segments
Point Fixing		At the point
Length fixing		In the middle of the object
Angle fixing		In the middle of the object
Fixed Dimension		
Dimension with Variable		Under the dimension line; peculiarities of displaying dimensions with variables are described in Section 7.2.5.1.1.

7.2.5.1.1. Peculiarities of displaying dimensions with variables

If the associative dimension has a variable, the name of this variable will be displayed in its dimension text (Fig. 7.2.5.1) in the mode of displaying relations and constraints (Fig. 7.2.4) The variable's name is displayed in parenthesis above the dimension line of the dimension. It is not printed out.

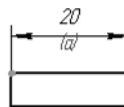


Fig. 7.2.4. Display of the variable's name in the dimension text

In case of controlling dimensions, not only the variable's name can be displayed, but also its value, if the value of the dimension's variable is set by a constant, expression or a negative number. Example of displaying a variable: $(a=b+10)$.

Only the variable's name is displayed in case of informational dimensions.

7.2.5.2. Degrees of Freedom



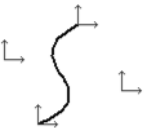
To display degrees of freedom symbols that graphical objects have, call the **Display Degrees of Freedom** command.

Upon calling the command, the degrees of freedom symbols will be displayed (see Table 7.2.3).

Tab. 7.2.3. Symbols indicating object degrees of freedom

Object	Displaying a maximum number of degrees of freedom	Comment
Point		
Line		The degrees of freedom for each line segment end are shown
Circle		The degrees of freedom for the center and the radius are shown
Circular Arc		The degrees of freedom for the center, radius and arc ends are shown
Ellipse		The degrees of freedom for the center, semi-axes and a rotational degree of freedom about the center are shown
Elliptical Arc		The degrees of freedom for the center, ends and a degree of freedom for the rotation about the center are shown

Tab. 7.2.3. Symbols indicating object degrees of freedom

Object	Displaying a maximum number of degrees of freedom	Comment
Spline		The degrees of freedom for ends and handles are shown

If there are constraints applied to the object depriving it of one or several degrees of freedom, then the symbols corresponding to the restricted degrees are not displayed. For example, Figure 7.2.5 shows step-by-step procedure of adding constraints to the lines that form a rectangle. At first, with the constraints *horizontality*, *verticality* and *points merging* added, every line end had two degrees of freedom, i.e. each of them could be moved in any direction with a mouse. Upon adding the constraint *fixed point* and setting two dimensions with the *fixed dimension* constraint, line ends lost all degrees of freedom. Now, the lengths of the given rectangle sides can only be controlled by modifying the values of dimensions. The lines of rectangle can not be moved with a mouse.

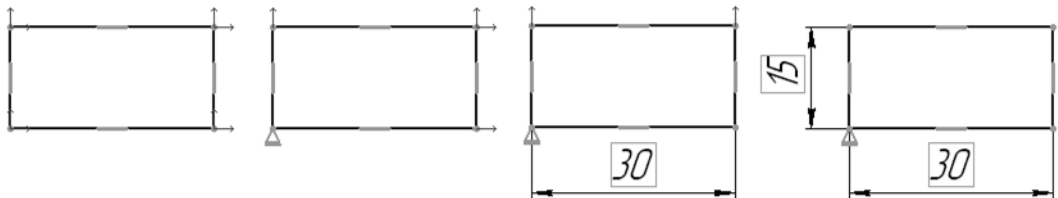


Fig. 7.2.5. Reducing object degrees of freedom

7.2.6. Peculiarities of executing some commands in parametric mode

7.2.6.1. Radial Dimension

When modifying a radial dimension on a shelf with several branches (see Section 3.3.3.2.1 on p. 1001), you should bear in mind the following:

If the dimension was set to objects in the parametric mode, editing—adding branches to new arcs or circles—should also be performed in the parametric mode, i.e. with the **Parametric mode** button clicked on the **Current status** bar. In the parametrization properties dialog (see Section 9.2.6.21 on p. 1830) associativity of dimensions option should be enabled

If the radial dimension is not associated with objects, the parametric mode should be disabled when adding branches.



7.2.6.2. Angular Dimension

When specifying angular dimension, you can use the axes of the absolute coordinate system as the sides of the angle (on the drawing—also the axes of the view's coordinate system).

In this case, an auxiliary straight line coinciding with a coordinate axis is created automatically in the document. This line has a *projection link* constraint. As long as this constraint is in effect, the line coincides with the axis, i.e. you cannot change its position. If you delete the *projection link* constraint, the line becomes available for editing.

7.2.6.3. Center mark

In the parametric mode, the associative designation of the center can be created using the **Center designation** command (see Section 3.3.7.14 on p. 1048) or when creating an axially symmetric object, for example, a circle with axes (see Section 3.2.5.1.1 on p. 910).

Enabling associativity of the center's designation is made in the parametrization settings dialog (see Section 9.2.6.21 on p. 1830).

The center mark has three forms: two axes, one axis and a symbolic designation (“cross”).

When creating associative center-marks, the following issues shall be accounted.

- ▼ As parameters of the base curve change, the associative axes are automatically rebuilt in such a way as to pass through the center of the curve and project beyond it by the specified distance.
- ▼ The axes can be extended/shortened by editing their defining points at the ends, however the edited axis stops “tracking” the curve extents in the respective direction.

To restore the axis “tracking” ability the curve extents in its direction, edit the Center mark by double clicking the mouse and switch the designation type to symbolic and back. Do these actions in parametric mode while the associative axes mode is enabled.

- ▼ The center point is not shown for associated designation of the center during editing displacing individual points (because center designation can not “come off” from the basic curve). If necessary, you can disable associative mode using **Show/Remove Relations and Constraints** command on the **Parametrization** panel.
- ▼ The Center mark only allows a non-associative dimension, therefore you cannot select axes when associative dimensions are enabled. If you would like a dimension to govern a curve center position, then it should be applied to the curve center itself.



7.2.6.4. Symmetry

Creating of related symmetrical objects can be performed by **Symmetry** command (see Section 3.4.4.3 on p. 1131).

To do this, run the command in the parametric mode and set the symmetry axis, specifying the object itself (not two of its points).

For example, to create the *symmetry* link when creating a symmetrical circle, click the **Base object selection** button on the Special control bar after calling the **Symmetry** command. Then, select with the cursor straight object.



7.2.6.5. auto centerline

Constraints *bisector*, *fixed length* and some other may be applied automatically to an auto centerline. Adding constraints *bisector* and *fixed length* can be disabled in the parametrization properties dialog (see Section 9.2.6.21 on p. 1830) In section, the case is considered when these limitations are **applied**.

When creating an auto centerline relative to two segments using **By Objects** method (see Section 3.3.7.16 on p. 1050), **Bisector** and other constraints are applied to the centerline. The types depend on the mutual arrangement of the objects, for which the symbols are created (fig. 7.2.6)

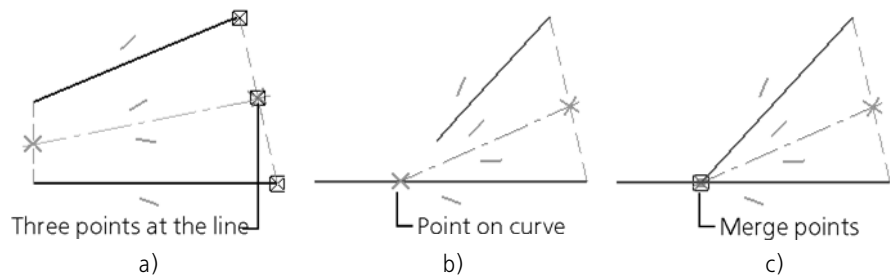


Fig. 7.2.6. Constraints, which were applied to auto centerline under different position of the segments

- ▼ If the segments are parallel and do not intersect each other, and the intersection point for their continuations does not lie on one of the segments, then **Three points on a straight line** limitations will be applied to its centerline in its ending points (Fig.7.2.6, a).
- ▼ If a segment intersects continuation of another segment, then **Point on a curve** limitation will be applied to the axis in the intersection point; in the opposite point— **Three points on a straight line** (Fig.7.2.6, b).
- ▼ If segments intersect each other, then **Merge points** limitation will be applied to the centerline in the intersection point; in the opposite point— **Three points on a straight line** (Fig.7.2.6, c).

In case when one or both end points of auto centerline are free from constraints, a restriction *fixed length* is applied. This occurs in the following cases:

- ▼ **By objects** relative to a Line Segment (see Fig. 3.3.79 on p. 1051),
- ▼ **By Selecting Boundaries** relative to a segment, if one boundary is not specified, see Fig. 3.3.83 on p. 1053 or both boundaries — see Fig. 3.3.82 on p. 1053,
- ▼ **By Selecting Boundaries** relative to two segment, if one or both boundaries are not specified — see Fig. 3.3.84 on p. 1054.

8. Print

8.1. Document printing

8.1.1. General Information about Document Printing

KOMPAS-3D is a Windows application and uses all capabilities of this operating system when working with output devices (printers and plotters).

You can print out a document, a selected part of the document, or a selected sheet of a multi-sheet document from the main window. In this case, all output settings are made in the print dialog (see Section 8.1.2 on p. 1631).

In order to print a multisheet document with variable sheet size, you can use the special print feature (see Section 8.1.3 on p. 1635).

If additional settings are needed prior to printing, for example, appropriate output scale should be selected, document page orientation should be modified and so on, the preview mode may be convenient (see Section 8.2 on p. 1639). The print preview mode allows simultaneous printing documents of different types, e.g. drawing and model.

When working in the preview mode, documents are output for printing in this mode. Editing documents in the print preview mode is not available.

Apart from standard print parameter settings, KOMPAS-3D offers a number of additional options to the user.

- ▼ automated fitting of the document sheet scale,
- ▼ different methods of placing sheets in the output field,
- ▼ printing of a specified sheet part only,
- ▼ layout and printing of several documents at a time,
- ▼ document object output controlling using filters.

Sometimes you have to print the documents with the same settings. In these cases, it is convenient to use print tasks (see Section 8.3.1 on p. 1655) and printer configurations (see Section 8.3.2 on p. 1659).

By default, to print out documents of all types KOMPAS-3D one printing device is used—that selected in Windows as the default one.

If required, you can set that certain devices be used for documents of certain types, by selecting the appropriate printing device configuration in the general printing properties dialog box (see Section 9.1.5.1 on p. 1700).

8.1.2. Order of Printing Documents



To print the current document, call the **File — Print...** command. The **Document printing** dialog will be displayed (Fig.8.1.1).

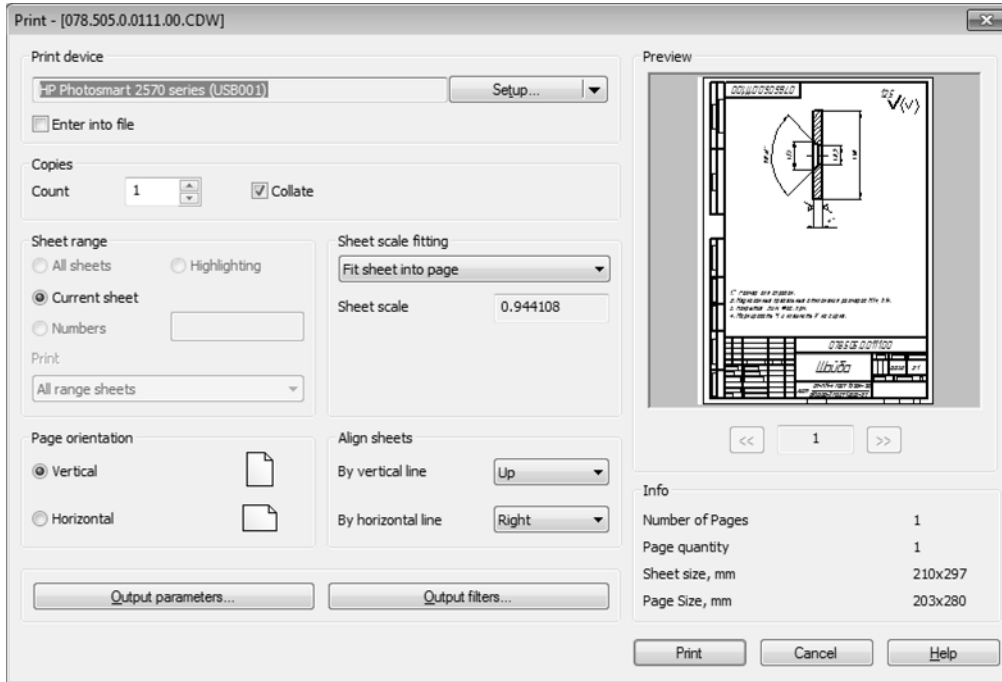


Fig. 8.1.1. Document print options dialog

The dialog's controls make it possible to perform standard setup of print parameters, for example, specify the number of copies, numbers of printed pages, and so on. If objects were selected in the document prior to calling the dialog, the **Selection** print option is automatically enabled. In this case, the document area inside the rectangle frame embracing all selected objects (in case of a text document, the selected text fragment) will be output for print.

Using the **Setup...** button, you can choose the required printer (printer or plotter) and set its parameters (see Section 8.1.2.1). Moreover, this button's menu makes it possible to use printer configurations (see Section 8.3.2 on p. 1659) to streamline print setup for document with similar parameters.



A device chosen for document printing will be used for all documents of the same type throughout the KOMPAS-3D working session by default.

If necessary, you can make additional document print settings, for example, choose a color use option in the process of document print output. For this purpose, call the additional setup dialog by clicking the **Output parameters...** button. (see Section 8.1.2.2 on p. 1634).

The **Output filters...** button allows to call the dialog for setting filters of document object output for print. Filters setup is described in Section 8.1.2.3 on p. 1634.



If anything was highlighted in a drawing or text document prior to calling the print dialog, the filter of the page's element output will be automatically disabled (frames, title blocks, etc.). If necessary, you can enable the filter. In this case, all page elements of a text document and only those parts of them within the frame embracing selected objects in a drawing, will be printed.

At the right part of the dialog there is preview window of print pages— paper sheets including “dead zones” the areas along the sheet margin on which the printer cannot print because of its design). One print page may contain one or several document sheets (depending on the setup). You can view print pages by using navigation keys in the bottom part of the preview window.

Once you have set the print options, click the **Print** button to begin document printing on paper.



If, when making settings, the print to file option was enabled in the dialog, a dialog will be displayed, in which you need to specify the file name and its folder location.



When you work with a graphic document you can print only its current page. For this purpose, the **Set print parameters and print page** context menu command is used. Upon calling this command, the described above Print Settings dialog will appear. Dialog control elements for multipage document setup will not be available.

Sometimes, additional document settings are needed prior to printing, for example, appropriate output scale should be selected, document page orientation should be modified and so on. In this case, the preview mode may be convenient (see Section 8.2 on p. 1639).



Note the following specifics when printing models.

Optimization of dimensions and symbols display is used for model display on the screen only and not for printing. When printing models legends in dimensions and symbols may be too small compared to the model itself. In this case disable the optimization (see Section 9.1.11.12 on p. 1736) and change the settings of dimensions and symbols in this model by increasing font height.

8.1.2.1. Printing Device Selection and Setup

KOMPAS-3D allows you to output documents to any external device which is supported by the operating system.

A plotter or printer is referred to as connected, or accessible, if its name is displayed in the list of available devices.



To obtain information about how to connect a printer or plotter, refer to the documentation on Windows OS and on the output device that you have.

To select the desired plotter/printer and set up its parameters, do the following.

1. In the document print options dialog (see Figures 8.1.1 on p. 1632 and 8.1.4) click the **Settings...** button. If, once the button has been clicked, there appears the message that settings will be saved for the current KOMPAS-3D working session only, click **OK**.
2. In the displayed standard **Print Setup** dialog, select the desired device name from the list of available devices.
A device chosen for document printing will be used for all documents of the same type throughout the KOMPAS-3D working session by default.
3. Set size and orientation of paper sheets. The paper size is selected from the list of sizes supported by the printer or plotter.
4. Choose the desired paper feeding method when printing.
5. If you need to run setup for various printing parameters (color of printing, print quality, etc.—the set depends on the device type), click the **Properties...** button to access the system dialog of the device driver.
6. Upon completing all required settings, click the **OK** button.

8.1.2.2. Additional output parameter settings

Additional document print output parameter settings are made in the dialog shown in Fig. 8.1.2. To call this dialog, press the **Output parameters...** button in the document print or special print options dialog (see Fig. 8.1.1 on p. 1632 and 8.1.4).

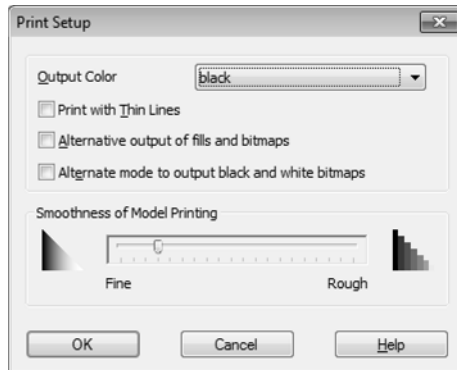


Fig. 8.1.2. Output Advanced Settings Dialog

The additional setup options dialog can be used to enable printing an image with thin lines, use alternative methods of printing fills and bitmaps, choose a color use option during printing, set the precision of printing.

Upon completing all required settings, click the **OK** button. To close the dialog without applying settings, press the **Cancel** button.

8.1.2.3. Print Filters Setup

Sometimes, a document needs to be printed in such a way that some layout objects or graphical objects are not printed. Filters are used for this. Print Filters Setup is done in the same-

name dialog (Fig.8.1.3). To call this dialog, press the **Output filters...** button in the document print or special print options dialog (see Fig. 8.1.1 on p. 1632 and 8.1.4). Application of output filters set up in the dialog does not change default system settings. The dialog controls are described in Section 9.1.5.2 on p. 1701.

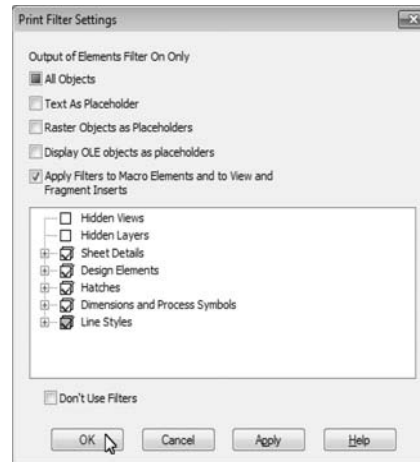


Fig. 8.1.3. Print Filters setup dialog

8.1.3. Special printing

Special printing is designed for the current document output to the printing devices that support changing page size in accordance with the output document's sheet size in the process of printing. For example, a single xps file is created with page sizes corresponding to pages sizes of the output document as a result of multipage document output through the special print mechanism to the virtual xps printer.

When printing fragments and models using the special print mechanism, a one-page document is created with the page size corresponding to dimensions of the output fragment or model image.



During normal printing of a multipage document with variable page format, depending on the set print page size, you will have to either divide document pages in several parts (based on the size of a print page) or group them on a single print page.



In order to print a document using the special print mechanism, call the **File — Special print...** command. The **Document special printing** dialog will be displayed (Fig.8.1.4).

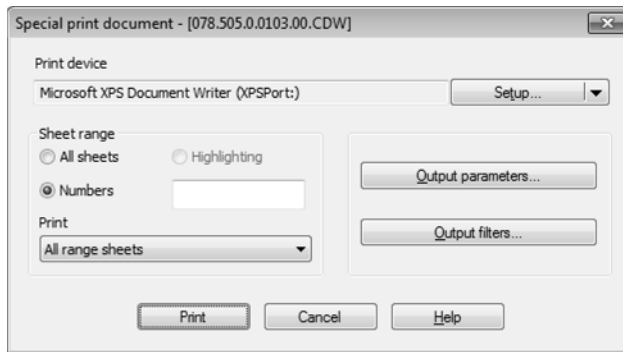


Fig. 8.1.4. Dialog for special print parameters setup

The dialog's control elements make it possible to setup the output page range, specify the numbers of printed pages, their output order (even, odd).

Using the **Setup...** button, you can choose the required printer and set its parameters (see Section 8.1.2.1 on p. 1633). Using the button's menu, you can use printer configurations (see Section 8.3.2 on p. 1659).

Additional document print settings are made in the additional settings dialog. You can call it by clicking the **Output parameters...** button. (see Section 8.1.2.2 on p. 1634).

The **Output filters...** button allows to call the dialog for setting filters of document object output for print. Filters setup is described in Section 8.1.2.3 on p. 1634.

Once you have set the print options, click the **Print** button to begin document printing on the chosen printer.



If a printer is selected for special printing that does not support changing page size during printing, the document will be output for printing in the same way as it would be output for normal printing — on pages with the size set by default for the printer.

8.1.4. Specifics of Outputting Documents to Vector Devices

There are certain special features of printing KOMPAS-3D documents on vector devices (for example, plotters).

- ▼ Documents to be printed to a vector device should be assigned to special layouts which use vector fonts. These layouts are stored in the system library *Vector.lyt*.

Layout change of current and new graphic documents is described in Section 3.5.2.1 on p. 1151; of current and new text documents in Section 4.1.5.2.

- ▼ Such graphical objects as TrueType fonts, solid color fills and bitmap images may not output correctly to a vector device. Therefore, a warning will be displayed when attempting to print a document containing the listed objects (Fig.8.1.5).

It lists the documents in which the following was found: TrueType fonts, bitmap images and fills, as well as object types (for example, dimensions, macro elements, etc.) containing TrueType fonts, bitmaps and fills.

You can disable the output of bitmap images and fills, or specify whether or not to print such documents, by clicking the respective button of the dialog.

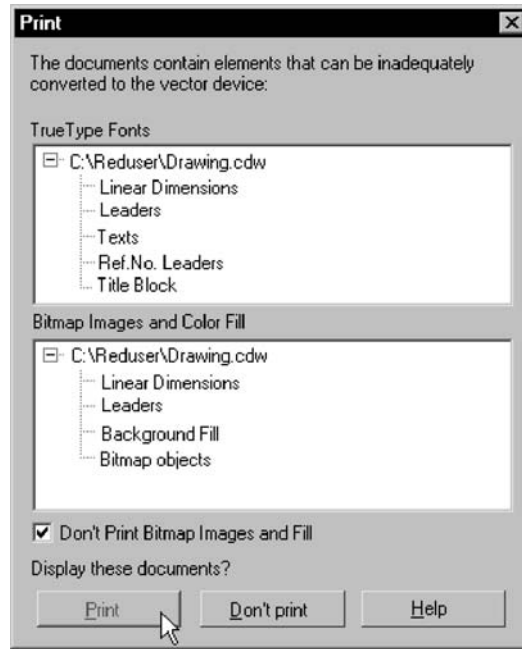


Fig. 8.1.5. Warning when printing to a vector device



When confirming the output of the documents listed in the dialog, remember that the enlisted objects are not guaranteed to be printed correctly.

8.2. Preview mode

8.2.1. General information on preview mode

8.2.1.1. Enabling and disabling print preview mode Print



Print Preview mode is a special operation mode of the KOMPAS-3D system.

To switch to this mode from standard system mode, call the **File — Print Preview** command or click the **Print Preview** button on the Standard bar.

If one document is opened in the main system window, it will be shown at once in the Preview mode.

If several documents are opened, the dialog will be displayed to select documents for printing. You may choose one or more documents by selecting them in the list. In addition, you may select closed documents for printing by clicking the button **From File...** The selected documents will be shown in the preview mode.

To disable displaying document selection dialog, and to display the current document in the preliminary view, click the **Preliminary view** button holding the <Shift> key.

If there is no open documents in the main system window, the **Print Preview** command is not displayed in the menu and the same-name button on the Standard bar is unavailable.



When working with a graphic document, you can preview both the whole document and its current page. For this purpose, use the **Send this page to preview** in the context menu.



You can perform the desired document display settings and print document in the print preview mode.

To start printing document, call the **File — Print** command. You can also click the **Print** button on the toolbar.



To finish work in Preview mode and return to the KOMPAS-3D standard mode, invoke the command **File— Close preview** or lick the mode symbol in the display window.

8.2.1.2. Print preview window interface

Preview mode has its own Main Menu, toolbar (Fig.8.2.1) and Property Bar (Fig.8.2.2).



Fig. 8.2.1. The main menu and toolbar of the preview window

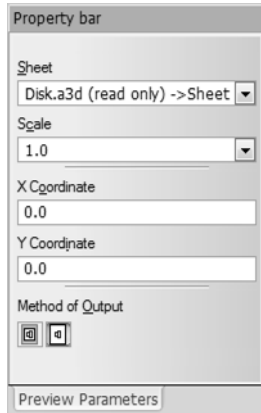


Fig. 8.2.2. Property bar



In this part of the Manual the words “menu”, “Toolbar” and “Property Bar” shall mean the Main Menu, Toolbar and Property Bar in Preview mode, unless otherwise indicated.

Some commands can be called from the context menu (Fig.8.2.3).

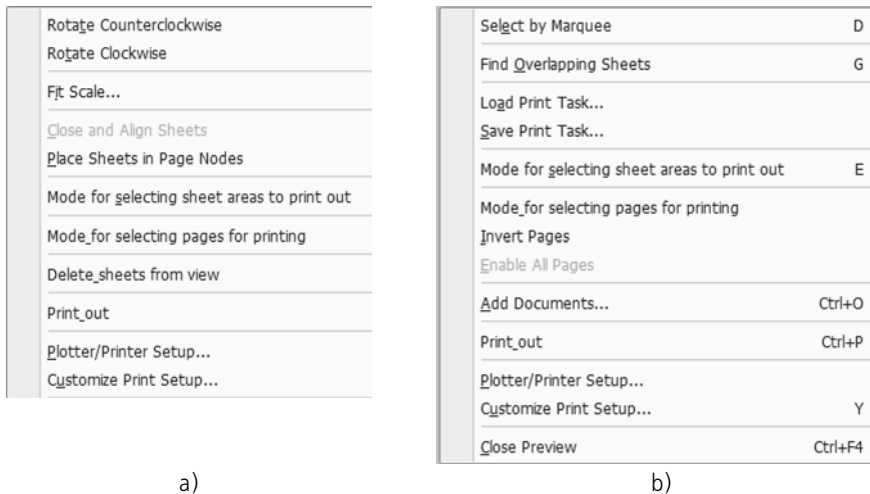


Fig. 8.2.3. Context menu in preview mode
 a) selected sheets; b) free space in the output field

In Preview mode, the printable area is schematically shown on the screen (one or several sheets of paper). It shows a realistic view of the selected document (or several selected documents).

If the paper size defined in the settings of the current printer is smaller then the document (or documents) image, the system automatically calculates the number of paper sheets required for the printout. In this case, the preview output field is divided by dotted lines into portions corresponding to the current paper size and orientation.

You may locate documents in the output field in the most convenient way (see Section 8.2.3).

The size of paper sheets including “dead zones” (the areas along the sheet margins on which the printer cannot print because of its design) and the required number of pages are displayed at the bottom part of the screen— in the Status line.

In this part of the Manual a paper sheet displayed in the output field is a **print page**, a sheet of a many- or single-sheeted document, selected for preview— a **document sheet**. For a fragment or model, the image dimensional rectangle shall serve as a document sheet.

8.2.1.2.1. Preview Scale

By default, the printable area is displayed so as to fully fit on the screen.

The current scale for displaying the output field is indicated in the field of the same name on the toolbar.

To change the display scale of the output field, the commands of the **Scale** submenu of the **View** menu are used.

If there are many documents (or the selected document contains several sheets), then the output field display scale becomes too small. This makes it inconvenient to position documents.

You can increase the display scale of the output field so as to make page borders and the printed image well visible on the screen.



To do this, call the **Zoom In** command or press the <+> key on the numeric keypad of the keyboard. The current scale will be increased by a factor of 1.2.



In this window, you can also zoom in on any region of the output field. To do this, call the **Zoom In by Marquee** command. After that, click with the mouse at the first corner location of the zoom area. Then move the cursor to get the desired marquee size. The ghost marquee box will be reflected on the screen.

After fixing the other corner of the box, the image will be magnified in such a way that the portion of the output area within the box fits to the entire window.

Additionally, you can define an arbitrary display scale. To do this, enter the desired value in the **Current Zoom** field on the toolbar and press the <Enter> key.



To reduce the scale, use the **Zoom Out** command or press the <-> key on the numeric keypad of the keyboard.

You can also return to one of the previous display scales of the output field. You do this using the **Previous** command. To advance to the next scale, call the **Next** command.



In order to return to the scale under which the entire output field is shown on the screen, click the button **Show All**.

When the display scale of the output field is increased, only a portion of this area appears on the screen. To view other portions of the output field without changing the scale, use the image moving command.



To call the command, press the **Move** button.



After that, the cursor shape will change: it will turn into a four-pointed arrow.

Move the cursor with the left mouse button pressed. The displayed area with the documents placed there will be scrolling along with the cursor movement. If the screen border is reached, and you need to move the output pane even further, release the mouse button, move the cursor

to a desired position, and then press the left mouse button again and continue scrolling the working field.

8.2.1.3. Adding and deleting document sheets



To add documents to a print preview, call the command **File— Add Documents...** In the opened dialog, select the required file name (or several file names) and press the **Open** button. The dialog will be displayed to select an option of adding document sheets (Fig.8.2.4).

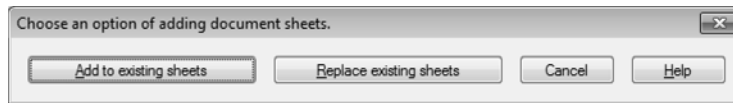


Fig. 8.2.4. The Dialogue to Select a Version to Add Document Sheets

Click the button **Add to Existing Sheets** to add sheets of loadable documents to the sheets of documents opened for preview. You may replace sheets of documents opened for preview with sheets of loadable documents. For this, click the button **Replace Existing Sheets**.

Note that all sheets of the selected documents are added to a print set.

If the “gap between sheets” option is enabled while setting output parameters, the added sheets will be shifted away from each other by the gap size.



To delete one or several sheets from the print preview, select them and call the **File — Delete Sheets from Preview** command.

The same sheet can be added to the print preview several times (for example, to print several areas of one sheet).



For quick copying of pages that are already available in the preview, choose the required pages and move them by the mouse keeping the *<Ctrl>* key pressed. In the process of copying, you can use snaps to page nodes and to other sheets' corners (see Section 8.2.3.1 on p. 1646).

8.2.1.3.1. Cancel printing of indicated pages

In case sheets (or a sheet) do not fit in the format which a connected printer or plotter can output, the system automatically positions an image on additional formats (pages).

You can control output of every page. To suppress some pages when printing, do the following.



1. Call the command **Service — Print Page Selection Mode**.

The system will enter the mode for specifying pages in the print area. This mode is indicated by displaying the pressed button **Print Page Selection Mode**. In this mode, no actions are taken regarding documents' layout.

2. Click the button on the page that needs to be blocked from printing. At the same time, the color in which the selected page is displayed on the screen will be changed. Re-clicking inside the page marked earlier re-enables its printing.
3. To exit the page-selection mode, recall the command **Print Page Selection Mode** or release the respective button.

8.2.2. Scaling document sheets

Print preview mode provides different opportunities of scaling document sheets.

When entering this mode and when adding sheets to the preview, automated fitting of the sheets scale is carried out (see Section 8.2.2.1).

When working in the preview mode in future, you will be able to customize page size (see Section 8.2.2.2 on p. 1644).

In addition, you may set the default print scale in the output options setting dialog (see Section 8.2.5.2).

You can also change print scale of the specific page. To do this, select the desired page and specify the required scale value in the scale control field located in the Properties bar. You can type the desired value manually or select it from the drop-down list.

8.2.2.1. Automated Fitting of Sheets Scale

Automated fitting of the document sheets scale is an automated rescaling of sheets to fit them to and place on the print pages.

Automated scale fitting is performed when you switch to the document viewing mode or add other documents to this mode.



If you have downloaded a print task file, no automated fitting is performed.

Enable automated fitting through the **Automated Fitting When Switching to Preview Mode** option available in the Print Setup Dialog (see Figure 8.2.6 on p. 1653).

The automated scale fitting comprises the following steps.

1. Check of sheet dimensions.

All dimensions shall comply with one of the following requirements:

- ▼ the same sheet dimensions,
- ▼ the respective sheet dimensions (height, width) are identical or multiple of each other.

If neither requirement is complied with, the automated fitting interrupts.

2. Calculation of sheet scales.
3. Check of the resulted scale.

Acceptable scale shall be limited by:

- ▼ maximum— default scale (preset during setup of the print parameters, see Section 8.2.5.2 on p. 1652);
- ▼ minimum— 0.94 scale of the default scale.

If the resulted scale is less than the minimum, the automated fitting interrupts.

If the resulted scale exceeds the maximum, the sheets will be set to default scale.

4. Automated Layout of Sheets on Print Pages.

The left lower corner of sheet 1 is placed in the output field point with coordinates 0.0.

Each succeeding sheet will be placed to the right from the previous one (with a gap, if preset) until there is a free space on the print page, when no free space is available the sheets will be placed to the left lower corner of the next print page's node.



During automated fitting of scale, sheet orientation remains unchanged.

When you add documents for viewing, their sheets are placed similar to succeeding sheets of a multi-sheeted document.

Automated fitting provides optimum results if:

- ▼ sheet dimensions correspond to the print page dimensions (for example, the same sheet and print page formats - A4),
 - ▼ you can select an acceptable scale to allow the sheet's horizontal and/or vertical dimension to be multiple of the corresponding print page dimension, i.e. a document can be printed on the whole number of pages (for example, an A3 sheet can be printed on two A4 pages).
 - ▼ you can select an acceptable scale to allow the sheet's horizontal and/or vertical dimension to be equal to the corresponding print page dimension, i.e. a whole number of sheets can be printed on one page (for example, an A1 sheet can be printed on eight A4 pages).
-



If the “gap between sheets” option was enabled during the output parameters setup, the gap will be taken into account during scale determination.

8.2.2.2. Fitting Sheet Scale



Sometimes, you need to fit the sheets in a certain number of pages; in this case, the image scale on the paper is not known in advance. To do this, select the sheets, and then enable the **Service— Fit Scale...** command.

After calling this command, the standard dialog appears to allow you to fit scale of sheets (Fig.8.2.5). The control elements of this dialog are represented in Table 8.2.1.

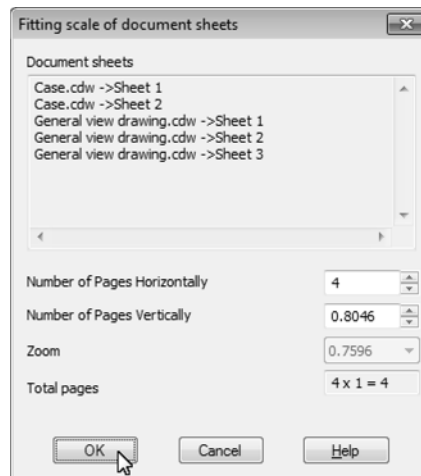


Fig. 8.2.5. Fit Document Scale Dialog

Tab. 8.2.1. Control Elements of the Fit Document Scale Dialog

Element	Description
Sheets of the Document	Reference field displaying number of sheets selected for scale fitting. Each sheet contains the document file name and its number in the document.
Number of Pages Horizontally, Number of Pages Vertically	Type in, set by the counter, or select from the list the desired value of any parameter. The 2nd parameter will be calculated in automated mode.
Scale	Scale Setup Field. You may not enter an arbitrary scale; it may be selected from the list only. In addition to standard values, it lists scales which were set earlier during work with this dialog. When you scale one sheet this field is available to preset a scale; however, while scaling several sheets the field is not available (and remains empty if the selected sheets have different original scales).
Total pages	The information field displaying the number of print pages with specified parameters.

The scaling point of the selected sheets is the left lower corner of their dimensional rectangle. If there is a gap between the sheets, it will also be scaled.



Sheets with different original scales will retain their 'own' scales after the scaling operation.



It is recommended to select neighboring sheets. Otherwise, the selected sheets may be overlapped by other sheets. To find overlapping sheets enable command **Service — Find Overlapping Sheets** (see Section 8.2.4.1 on p. 1650).

8.2.3. Locating Documents in the Output Field

There are the following provisions to position a document in the output field:

- ▼ movement
- ▼ Rotation,
- ▼ Scaling.

To change position of a sheet (sheets) in the output field, click on its image, select its name from the **Sheet** list on the Property bar or use the `<Tab>` key. It is displayed surrounded by a green outline frame.

To select several sheets, specify them with the `<Ctrl>` key pressed.



You can also use the cutting or enclosing marquee to select several sheets. This is possible through the **Service— Select by Marquee** command.



To select all sheets enable the **Service— Select All** command.

Sections 8.2.3.1 and 8.2.3.2 are devoted to locating one current document and Section 8.2.3.3— to locating several documents at a time.

8.2.3.1. Moving Documents

A document sheet can be moved within the output field as follows:

- ▼ Using arrow keys
- ▼ By specifying base image point coordinates
- ▼ With the mouse:
 - ▼ Arbitrary
 - ▼ By snapping to the print page node,
 - ▼ By snapping to a corner of another sheet.

When moving the sheet by the arrow keys, a single stroke on an arrow key moves the image by one step. To change the move step size, enter or select one from the **Cursor's Current Step** list on the toolbar.

Once the desired position of the image format frame is achieved using arrow keys, fix it by pressing the `<Enter>` key. The image will be redrawn according to the new position of the sheet in the output field.

You can define the exact position of the sheet by entering its base point coordinates in the respective fields on the Property Bar. The coordinate system origin is the lower left corner of the printable page, while the sheet base point is its lower left corner. The symbol for the coordinate axes is positioned at this point. The symbol is used only for working convenience and is not printed on paper.

Note that each coordinate input must be confirmed by pressing the `<Enter>` key.



The **X** and **Y** fields allow entering only positive values, since a negative offset would make some portion of the sheet outside of the printing area.

Arbitrary sheet placement in the print field is convenient to use when you need to print a document with large margins. Positioning by snapping to the corners, on the other hand, helps save paper by printing documents next to each other or as close as possible to the page borders. Document movement with the mouse (arbitrary and with snapping) is detailed below.

8.2.3.1.1. Arbitrary Movement

To move the sheet in the printable area, complete the following steps.

1. Mark the sheet to be moved as current.
2. Place the cursor within the document limits.



The cursor appearance will change.

3. Click the left mouse button, hold it down and drag the mouse. The sheet outline frame will be moved across the output field.
4. When the desired location for the outline frame is reached, release the mouse button.

The image will be redrawn according to the new position of the sheet in the output field. Additional pages will automatically appear in the printable field if the document extends outside the current printable area margins.

8.2.3.1.2. Movement with Snap to Page Nodes

When one of the corners of the moveable sheet outline frame approaches the corner of any page constituting the output field, a marker appears as a small circle in the sheet corner. If you release the mouse button when the marker is displayed, the appropriate sheet corner will snap to a page node (other sheet corners may not match precisely with page nodes if the document size is not a multiple of page size).

The relative distance value between the node and the corner of the sheet outline frame at which the snap mark appears, can be modified in the Print Setup dialog (see Section 8.2.5.2 on p. 1652). You can disable snapping of the document to page corners in the same dialog.



Snapping to the page nodes can be also disabled and enabled by the **Snap to Page Nodes** button on the toolbar.

8.2.3.1.3. Movement with Snap to Corners of Other Documents

If several documents were selected for the preview, then you can move them across the output field, snapping sheet corners to each other.



Unlike the order of operations described above, such relocation is done by the mouse, but, before moving, place the cursor closer to the sheet corner rather than at the center, so that the cursor is displayed as a boxed corner (the cursor orientation depends on near what corner it was fixed).

Click the left mouse button and move the sheet outline frame in the output field.



When the frame corner beyond which you move an image approaches the frame corner of another sheet's outline frame, a marker appears as a small square inside the cursor frame. If you

release the mouse button while the marker is displayed on the screen, the respective sheet corner will snap to the corner of another document sheet (in this case, other corners of these documents may not match if the sheet side dimensions are different).

The relative distance value between sheet corners at which the snap marker appears can be changed in the output options setting dialog (see Section 8.2.5.2 on p. 1652). The same dialog can be used to disable gap using in the snapping sheet corners to each other and to specify gap value.



You can also enable and disable using a gap when snapping to sheet corners using the **Keep gap between sheets** button on the toolbar.



When an sheet is “drag-and-dropped” over the corner, it is not snapped to the print page node.

8.2.3.2. Rotate Sheet



To use paper more efficiently, you sometimes may need to rotate the printable sheet.



Rotating a document is carried out with the commands **Rotate Counter Clockwise** and **Rotate Clockwise** of the **Service** menu.

8.2.3.3. Placing Several Sheets

You can place a few selected sheets to the output field using methods presented in Table 8.2.2.





It is recommended to select neighboring sheets. Otherwise, the selected sheets may be overlapped by other sheets. To find overlapping sheets enable command **Service — Find Overlapping Sheets** (see Section 8.2.4.1 on p. 1650).

Tab. 8.2.2. Methods of Placing Sheets in the Output Field

Method	Description
Rotate Sheet	Rotating, moving and scaling operations are the same for the several selected or only one selected sheet (see Sections 8.2.2, 8.2.3.1, and 8.2.3.2). The base point of the selected sheets is the left lower corner of their dimensional rectangle.
Move Sheet	
Scale Sheet	

Tab. 8.2.2. Methods of Placing Sheets in the Output Field

Method	Description
	<p>Closure and Alignment of Sheets This method allows changing position of the selected sheets in the following way: firstly, the sheets move to the left up to the boarder of their dimension rectangular or the nearest selected sheet, and then move down. To move sheets by this method enable the Service — Closure and Alignment of Sheets command¹.</p> <p>If, when setting up output parameters (see Section 8.2.5.2 on p. 1652) the “add gap between sheets” option is enabled, the gap will make sheets to move towards each other. During such movement the sheet orientation remains unchanged.</p>
	<p>Placing Sheets in Corners This method allows changing position of the selected sheets in the following way: the sheet left lower corners coincide with the left lower nodes of the neighboring print pages. To do this, enable the Service — Place Sheets in Page Nodes command¹. The sheet will move to the nearest page node provided that this sheet's left lower corner is in the “sensitivity area” of the page node.</p> <p>The 'sensitivity' radius is preset in the output setup dialog.</p>

¹ This command may result in overlapping of sheets. This depends on relation between the sheet dimensions and the print page, as well as the original position of sheets in the output field. To find overlapping sheets enable command **Find Overlapping Sheets**.

8.2.3.4. Examples of Placing Sheets in the Output Field

Example 1: placing sheets whose dimensions correspond to or multiple of the print page dimensions (for example, the print page format is A4, sheet formats are A4 and A3).

1. Select a list whose disunions are much the same as the print page dimensions.
2. Fit the scale of the selected sheet with the aid of the **Fit Scale** command... (see Section 8.2.2.2).
3. Set the resulted scale for the rest sheets.
4. Place sheets in the output field as required.
5. Select all sheets.



6. Close sheets to each other and align them with the aid of the **Closure and Alignment of Sheets**.



7. Locate sheets in the print page nodes using the **Place Sheets in Page Nodes** command.



8. To check sheets for overlapping enable the **Find Overlapping Sheets** command.

Example 2: placing sheets whose dimensions are less than the print page dimensions (for example, a few A4 sheets are printed on the A1 page).

1. Locate close all sheets on the print page, avoid overlapping.
2. Select all sheets.
3. Close sheets to each other and align them with the aid of the **Closure and Alignment of Sheets**.
4. Fit the scale of the selected sheets with the aid of the **Fit Scale** command...

8.2.4. Advanced capabilities

8.2.4.1. Search for Overlapping Sheets



You can enable search for overlapping sheets through the **Service— Find Overlapping Sheets** command.

Once enabled, this option checks all sheets added to the view mode for overlapping.

- ▼ Detected overlapped sheets are highlighted with green frames. Sheets which fully overlap each other are highlighted with a red frame.
- ▼ If no overlapping sheets are found, the respective message will be displayed on the screen.

8.2.4.2. Print sheet area

You can print the whole document sheet or its area bounded by a rectangle of an arbitrary size. Use the **Output Method** group of switches of the Properties bar to select the required printing option.



Output Current Sheet in Full Size switch activation allows to print the whole selected document sheet. In this case, it is fully displayed in the output field.



Output Part of Current Sheet switch activation allows to print the selected document sheet area. In this case, only selected area is displayed in the output field.



To set the borders of print areas of a document's sheets, use the **Service — Sheet printable area setup** mode command.

Once the command has been called, the **Area selection method** group of switches and current sheet print area control fields will appear in the Properties Bar. Entering values in these fields allows to define area size and position. The set of fields depends on the method of specifying sheet printing area.



There are two methods to specify area:

- ▼ specifying area size and position,
- ▼ by specifying indents from sheet margins.



To select the required method you need to activate the corresponding switch.

By default, dimensions of the page printing area frame match its extents. In order to change size or position of the frame, enter the required values in the printable area control fields (you can either enter numbers or expressions for their calculation). In addition, you can move the sides or corners of the frame and also the whole frame using the mouse. In this case, the values in the fields will change automatically.

If the document contains several pages, you can specify printing area of each page without interrupting the command. To select the required page, click on its image, or specify its name in the **Sheet** list in the Properties bar or use *<Tab>* key.



The current page is fully displayed in the output field. For another pages, only areas set for printing are displayed.



To complete the printable area setup, call the **Printable area setup command** again or press the *<Esc>* button.



After this, only those sheet areas will be displayed in the output field, that are set for printing. The **Output the selected area of the current sheet** switch will be enabled in the **Output method** group of switches of the Property bar. If necessary, you can change the way of printing of each page.



You can select only one printing area on one page.

If you need to print several areas of the same page, perform the following actions:

- ▼ create the required number of page copies by moving it with *<Ctrl>* key pressed,
- ▼ call the **Printed sheet area selection mode** Command
- ▼ select the required areas on the obtained page copies.



A document sheet area set for printing can be moved and rotated in the output area, or scaled just like the whole sheet.



When working with graphical documents, you can specify print area prior to going to the preview mode. For this purpose, zoom out so that the window displays only the part of the image that should be printed.

8.2.4.3. Printing the selected part of a document



You can limit the image section for output by specifying objects in the document that should be printed. For this purpose, select the objects you need (in a text document, a text fragment) and call the **Send selection to preview** command from the **Preview** button's menu.

After this, the document will be opened in the preview mode and also:

- ▼ for graphic documents and models:
 - ▼ output of the specified area is automatically enabled (in the **Output method** group of the Property bar, the **Output selection of the current page** is enabled, see Section 8.2.4.2),
 - ▼ the print area represents a dimensional rectangle with selected objects, i.e. all other objects that are fully or partly embraced by the rectangle will be printed further to the selected ones.
- ▼ for text documents: the selected fragment is positioned in the upper part of the page irrespective of its position within the text;

- ▼ additionally, for drawings and text documents:
 - ▼ filters' settings will be changed automatically: output of page elements (frame, title block, and so on) is disabled. (if necessary, you can enable printing of these elements – see Section 8.2.5.3 on p. 1653; in this case, the whole page's elements will be printed in the text document; in a drawing, only those sections of them will be printed, which are fall within the dimensional rectangle embracing the selected objects);
- ▼ additionally, for multipage drawings:
 - ▼ since only one print area can be specified in one page, several print areas are created – one for each page within the dimensional rectangle embracing the selected objects or intersecting it,
 - ▼ pages outside the dimensional rectangle embracing the selected objects and not intersecting it are not output to the preview.

When working BOM, the **Send selection to preview** command is not available.

8.2.5. Print preview settings

8.2.5.1. Selection and Setup of the required Printing Device

KOMPAS-3D allows you to output documents to any external device which is supported by the operating system.

A plotter or printer is referred to as connected, or accessible, if its name is displayed in the list of available devices.



To obtain information about how to connect a printer or plotter, refer to the documentation on Windows OS and on the output device that you have.



To select the required plotter/printer for printing from the preview mode, do the following.

1. Call the command **File — Plotter/Printer Setup...**
2. Select the required printing device and customize print parameters in the appeared on the screen dialog as it is described in Section 8.1.2.1 on p. 1633.

The selected device will be used by default for all type document printing from the preview mode in the current KOMPAS-3D working session.



You can also select and setup a printer/plotter for printing in the preview mode in advance when working in the KOMPAS-3D main window. To do this, call the command **File — Print Preview Settings — Plotter/Printer Setup...**

8.2.5.2. Print Parameter Setup



You can set up the parameters of document printing in the preview mode in the dialog shown in Fig. 8.2.6. This dialog appears on the screen after calling the **File — Print Setup...** command.

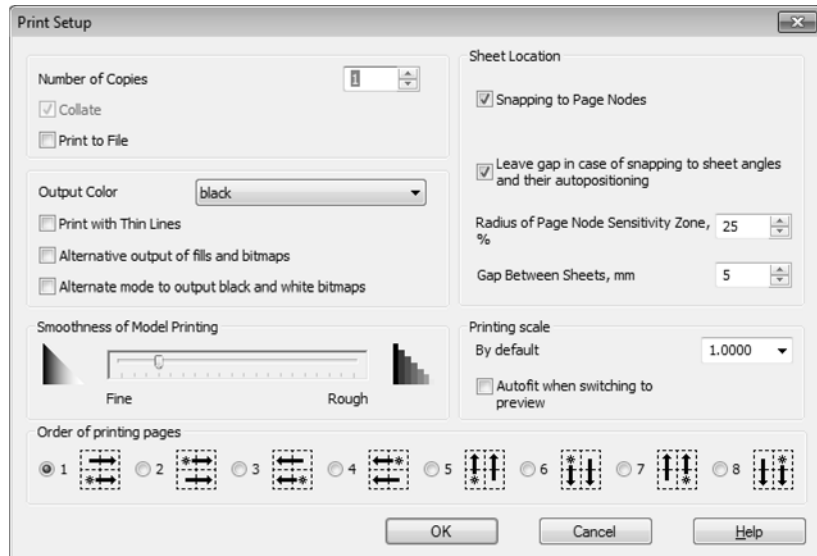


Fig. 8.2.6. Output Settings Dialog

Control elements of the dialog make it possible to set the number of document copies for print, page order and page print order, method of printing objects of different types, option of using colors for printing, print scale and precision.

Upon completing all required settings, click the **OK** button.

Such parameters as **Snapping to the Page Nodes** and **Allow Gap between Sheets** can be managed without dialog calling— using the buttons on the toolbar.



If the output color was changed in the dialog, and also, if the **Print with Thin Lines** option was toggled on or off, the image will be redrawn. Modifying the default scale in no way affects the open documents— the specified value will be applied only to the newly viewed documents.



If print to file was set, then, after calling the **Print** command, a dialog will be displayed, in which you need to specify the filename and its folder location.



You can setup the parameters of document printing in the preview mode in advance when working in the KOMPAS-3D main window. To do this, call the command **File— Print Preview Settings— Print Setup....**

8.2.5.3. Setup of Print Filters

If a document needs to be printed in such a way that some layout objects or graphical objects are not printed, use print filters.



In order to set filters for printing documents in the preview mode, call the **Print filters...** command in the **Service** menu of the preview window. Filter setup dialog will appear on the screen

(Fig.8.2.7). The dialog controls are described in Section 9.1.5.2 on p. 1701.

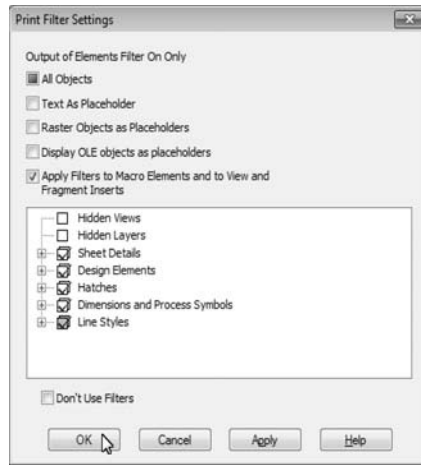


Fig. 8.2.7. Print Filters setup dialog

You can apply the created filters by closing the dialog or leaving it open. In the first case, click the **OK** button; in the second case, click the **Apply** button.



Output filters applied in the preview mode remain applicable only until you finish working in the mode.

8.3. Print Task. Printer configurations

8.3.1. Print Task

If you have to print the same documents with the same settings, it is convenient to use print tasks.

A print task is a file to which you write sheet numbers and names of document files selected for printing, settings of their location in the output field, output options settings and output device data.

A print task file has the *pjd* extension.

A task file is written in text format so that, if necessary, it can be opened and edited with any text editor, e.g., with Notepad, included with standard Windows programs.

When a print task file is loaded, document sheets written to it are displayed in the preview mode with saved settings.

8.3.1.1. Saving Print Task



In order to save a print task, call the command **File— Save Print Task...** in the Preview mode. Windows-standard File Save dialog will be displayed. Enter the file name and press **Save** button. By default, the task file is assigned the name “Printing task”.

Each new task is saved in a separate file. For this, the following data is saved:

- ▼ Relative and absolute paths to the document files selected for printing,
- ▼ Numbers and dimensions of document sheets selected for printing,
- ▼ The selected printing device and its settings,
- ▼ Location, orientation and scale of document sheets specified in the preview mode,
- ▼ page printability indicator of an output device set by **Service — Page selection for print mode** command,
- ▼ Filter settings,
- ▼ Output options settings, except for the options **Number of Copies**, **Default Printing Scale** and **Remember List of N Print Tasks**.



To enable movement of folders with drawing files and a print task file to other folders without impairing relevance of a task file, place a print task file next to the drawing files (in one folder) or in a parent folder.

You may write only those documents to a task file which have been saved at least once. If documents selected for printing include any that have never been saved (without a name), a message will be displayed when the print task is saved, that they cannot be written to the task file.

Click the **Yes** button to save the task and skip these documents. Press the **No** button to cancel saving the task.

If none of the documents selected for printing has ever been saved, the print task cannot be saved. When you attempt to save the task, a warning message will be displayed.

8.3.1.2. Loading Print Task



In order to load a print task, call the command **File — Load Print Task...** in the Preview Mode. The standard File Open dialog will appear on the screen. Select a name of the required task file and click the **Open** button.



If you have recently saved or loaded a printing task, you can select its name from the task list. This list is displayed:

- ▼ in the preview window— in the **File** menu,
- ▼ in main window— in the **File— Print Task** menu.

By default, the list contains nine recently saved or loaded printing tasks. The number of tasks in the list can be customized (see Section 9.1.5.1 on p. 1700).

A dialog will be displayed to choose an option of adding document sheets (Fig. 8.3.1).

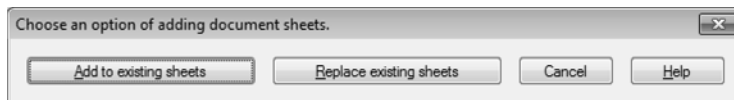


Fig. 8.3.1. The Dialogue to Select a Version to Add Document Sheets

Click the button **Add to Existing Sheets**, to add sheets of task documents to sheets of documents opened for preview. By doing so, sheets of task documents will be placed after sheets of documents opened for preview. Relative location of sheets will be saved.

If sheets are added, the printable page flag saved in the task for an output device, as set with the **Service — Page selection for print mode** command, will be ignored.



Current output settings will change subject to the loaded task.

Click the **Replace Existing Sheets** button to replace sheets of documents opened for preview with the sheets of task documents.

The **Cancel** button allows you to cancel loading a print task.



To load a print task from the main program window, call the command **File — Print Task — Load...** or select the name of the required file from the task list. In this case, the dialog **Select the Option of Adding Document Sheets** will not be displayed. You will automatically switch to the preview mode, in which document sheets of the task will be displayed.

If the current output parameters, printer or filter settings differ from those saved in the print task, the print task' settings and parameters apply if you change sheets; if you add sheet the following message will be displayed8.3.2.

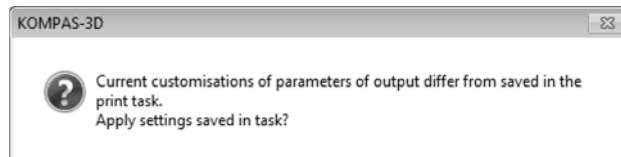


Fig. 8.3.2. Notification that the current output parameters differ from those, saved in the print task

Button **Yes** allows you to change the current parameters and settings for those contained in the print task.

Button **No** allows using the current parameters and settings.

After loading of the printing task in the preview window, you can work as usual: change current settings, delete or move document sheets, etc.



The performed changes are not saved in the file of the printing task. You may save current settings to a new task file or overwrite the existing one, if necessary.

8.3.1.2.1. Possible Problems when Loading a Task

When loading a print task, you may encounter various problems.

If a task file contains errors and cannot be loaded, there will be displayed the message **The print task file contains errors and cannot be loaded**.

If, when loading a print task, you encounter problems that do not prevent it from proceeding, the Information Window will be displayed on loading completion (Fig. 8.3.3).

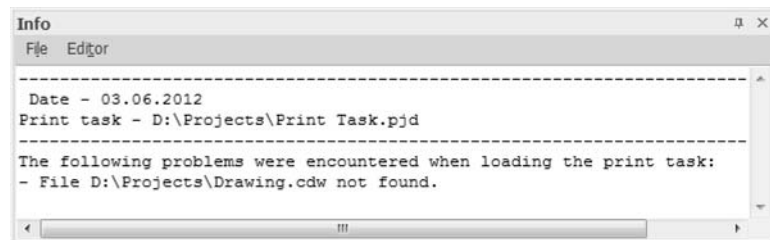


Fig. 8.3.3. Information Window

The Information Window has its own menu (see Tables 3.8.1 on p. 1264 and 3.8.2 on p. 1265). The window displays:

- ▼ The current date,
- ▼ The complete name of a print task file,
- ▼ The list of problems encountered at loading.

When loading a print task, the following problems may occur.

- ▼ **Output Device Data missing in Print Task. The current output device will be used.** This message appears in the information window if the print task has no output device data, which is specified in the **Print Setup** dialog. In this case, the output device will be the current or default one.

- ▼ **The output device specified in the print task does not exist in the system. The current output device will be used.** This message appears in the information window if the system has no output device saved in the print task. In this case, the current output device or that assigned by default is used to print. The printable page flag saved in the task for an output device, as set using the command **Page selection for print mode**, is ignored.
- ▼ **Page Parameters missing in Print Task. Output device settings will be used.** This message appears in the information window if the print task has no page parameters (size and orientation), which are specified in the **Print Setup** dialog. In this case, the output device settings are used when the task is loaded.
- ▼ **Print Filter Settings missing in Print Task. Current filter settings will be used.** This message appears in the information window if the print task has no output filter settings, which are specified in the **Set Print Filters** dialog. In this case, current filter settings are used when the task is loaded.
- ▼ **Some Filter settings are missing in Print Task. Current settings will be left for these filters.** This message appears in the information window if the print task has no settings for some output filters, which are specified in the **Set Print Filters** dialog. In this case the current settings of the filters, for which the settings are missing in the task, are used when the task is loaded.
- ▼ **Output Parameters missing in Print Task. Current output parameter settings will be used.** This message appears in the information window if the print task has no output options data, which is specified in the **Set Print Options** dialog. In this case, the current settings of these options are used when the task is loaded.
- ▼ **Some Output Parameters missing in Print Task. For these parameters the current values will remain unchanged.** This message appears in the information window if the print task has no data about some output options, which is specified in the **Set Print Options** dialog. In this case the current settings of the parameters, the data about which are missing in the task, are used when the task is loaded.
- ▼ **Sheet <sheet number> missing in file complete file name.** This message appears in the information window if one of the sheets of the documents written to the task is not found when the task is loaded. In this case, the task is loaded without this sheet. If all sheets of all documents written to the task are missing, loading of the task file is interrupted.
- ▼ **Printing area for sheet <sheet number> is out of limits, file <complete file name>.** This message appears in the information window if only a part of a sheet is specified for printing, and the print area borders fall outside the current sheet size. In this case, the task is loaded without this sheet. If this error occurs in all sheets of all documents written to the task, loading of the task file is interrupted.
- ▼ **File <complete file name> not found.** This message appears in the information window if one of the documents written to the task is not found when the task is loaded. In this case, the task is loaded without this document. If all documents written to the task are missing, loading of the task file is interrupted.
- ▼ **Cannot open the <fully specified file name> file.** This message appears in the information window if one of the documents written to the task is found but cannot be opened. In this case, the task is loaded without this document. If all documents written to the task cannot be opened, loading of the task file is interrupted.

- ▼ **Sheet dimensions, <sheet number>, file <full file name> have changed.** This message appears in the information window if the sheet dimensions written in the task differ from their current dimensions. Upon uploading the task the sheets are displayed with their current numbers. The sheets' position in the output field remains unchanged.



If loading of the print task is called from the main program window and all documents written to the task contain errors, or none of these documents is found, loading of the task file will be interrupted. The message will be displayed: **When processing the selected print task file, failed to produce output. For more details, enter the Preview mode and load this file.**

8.3.2. Printer configurations

You can use printer configurations for the following purposes:

- ▼ quick switching between printers and changing their parameters,
- ▼ set a default printer and document print parameters in preview mode,
- ▼ set default printers and print parameters for different types of documents in the main program window.

Printer configuration is a file to which current printing parameters (paper size and feeding, page layout) and printer information are saved.

A task file has *pdc* extension.

You can save the configuration file for any available printer (see Section 8.3.2.1). You can create several configuration files for a single printer.

A task file is written in text format so that, if necessary, it can be opened and edited with any text editor, e.g., with Notepad, included with standard Windows programs.

When loading a configuration, the current printer will be replaced by the printer specified in the configuration file. Print parameter values will be replaced by those from the configuration file (see Section 8.3.2.2 on p. 1661).

You can choose configurations that will be used by default when printing documents from the main window and in the preview mode (see Section 9.1.5.1 on p. 1700). In this case, the default printers will be those specified in the files.

If no default configuration file is selected, or the selected file is not found, the default Windows printer will be used by default.



You can specify a different printer and set its parameters when setting up document output for printing in the main window (see Section 8.1.2.1 on p. 1633) and in the preview mode (see Section 8.2.5.1 on p. 1652). These settings are retained during the working session.

8.3.2.1. Save Configuration

Each new configuration is saved in a separate file. The following data is saved to the file:

- ▼ current printing device name and parameters,
- ▼ the paper size and feeding method,
- ▼ print page orientation.



For convenience, we recommend to give such names to configuration files that not only reflect the names of current printers but also print parameters that had been set up (paper size and page orientation). You can also name files taking account of the purpose of configurations saved in them, for example, using document type names for which they are designed.

You can save a printer configuration using one of the following methods.

Method 1

1. Open a KOMPAS-document.
2. Call the **File — Print...** command. or the **File — Special print...** command. Print setup dialog or special print setup dialog will appear on the screen (see Fig. 8.1.1 on p. 1632 and Fig. 8.1.4).
3. Make sure that the required printer is used and the required parameters are set up (when setting up print parameters, different printers and print parameters can be set for different document types).
4. Call the **Save plotter/printer configuration...** command using the **Setup...** button. (menu is called by clicking the mouse on the triangle to the right of the button). Windows-standard File Save dialog will be displayed.
5. Enter the file name and press **Save** button. By default, the configuration file name is the name of the current printing device.

You can change print parameters or save them to a new task file or overwrite the existing one, if necessary.

6. Click the **Cancel** button to close the print setup dialog.

Method 2



1. Call the **File — Print Preview Settings — Printer/Plotter Setup** command. The view settings dialog will be displayed.
2. Make sure that the required printer is used and the required parameters are set. After that close the dialog.
3. Call the **File — Preview settings — Save Printer/Plotter configuration...** command. Windows-standard File Save dialog will be displayed.
4. Enter the file name and press **Save** button. By default, the configuration file name is the name of the current printing device.

Method 3



1. Enter the print preview mode by calling the **File — Print Preview** command or by clicking the **Print Preview** button on the Standard bar.



2. Call the command **File — Plotter/Printer Setup**. The view settings dialog will be displayed.
3. Make sure that the required printer is used and the required parameters are set. After that close the dialog.
4. Call the **File — Printer/Plotter configuration — Save Printer/Plotter configuration...** command. Windows-standard File Save dialog will be displayed.

5. Enter the file name and press **Save** button. By default, the configuration file name is the name of the current printing device.

8.3.2.2. Loading a configuration

You can load printer configuration when setting up document printing in the main window or in the preview window.

Loading a document print configuration from the main window is performed as follows.

1. Activate the desired document.
2. Call the **File — Print...** command. or the **File — Special print...** command. Print setup dialog or special print setup dialog will appear on the screen (see Fig. 8.1.1 on p. 1632 and Fig. 8.1.4).
3. Call the **Load plotter/printer configuration...** command using the **Setup...** button. (menu is called by clicking the mouse on the triangle to the right of the button). Windows-standard file opening dialog will be displayed.
4. Specify the desired task file and click the **Open** button.



If you saved or loaded the configuration recently, you can load it by selecting it from the list of configurations in the **Setup...** button menu. The list contains configurations that were saved or loaded for documents of the same type.

By default, the list contains nine recently saved or loaded task files. The number of objects in the list can be customized (see Section 9.1.5.1 on p. 1700).

5. If you need to print a document, click the **Print** button. If you do not need to print the document at the moment, click **Cancel**.



Note that data from the configuration file selected for document printing will be used for all documents of the same type.

You can load a configuration for document printing in the preview mode when working both in the main window and in the preview mode window. To ungroup, do the following.

1. Call the **Load plotter/printer configuration...** command in the **File — Preview settings** menu in the main window or the **File — Plotter/printer configuration** menu in the preview mode.
Windows-standard file opening dialog will be displayed.
2. Specify the desired task file and click the **Open** button.



You can select a configuration to use from the list of recently saved or loaded configurations located in the **File — Preview settings** menu in the main window or the **File — Plotter/printer configuration** menu in the preview mode.

When configuration file is loaded, the current printing device is replaced with the printing device specified in the file, paper size and page orientation parameters get values from the file. Any other print parameters remain unchanged. These parameters should be set up manually.

The data loaded from the configuration file are stored throughout the KOMPAS-3D working session.

You can change the values of parameters obtained from the configuration file or choose another printer. These changes are not saved in a task file. You may save current settings to a new task file or overwrite the existing one, if necessary.

8.3.2.2.1. Possible configuration loading problems

When loading a configurations, the following problems may occur.

- ▼ Printer specified in the configuration file cannot be applied. In this case, the following message is displayed: **Output device specified in the configuration file is not present in the system. Current output device will be used.** Current output device and print parameters remain unchanged.
- ▼ It is not possible to set the paper size specified in the configuration. In this case, the following message will be displayed: **It is impossible to set the paper size set in the configuration file.** The current printer will be replaced by the printer specified in the configuration file, page orientation will be assigned the value from the configuration file, paper size will be assigned default paper size values of the printer specified in the configuration file.
- ▼ It is impossible to load data from the configuration file (the specified file is not a configuration file, data is missing in the file or data is invalid and so on). In this case, the following message will appear on the screen: **The specified file is corrupt or is not an output device configuration file.** Current output device and print parameters remain unchanged.

9. KOMPAS-3D Settings

9.1. System Parameters

9.1.1. General System Setup Information

The KOMPAS-3D system setup is made on the **System** tab of the dialog called with **Service — Parameters...** command (Fig. 9.1.1).

In the left part of the tab there is the object setup list represented as a “tree” They are grouped by sections according to their purpose. After selecting an item in the left side, control elements will appear in the right side of the tab.

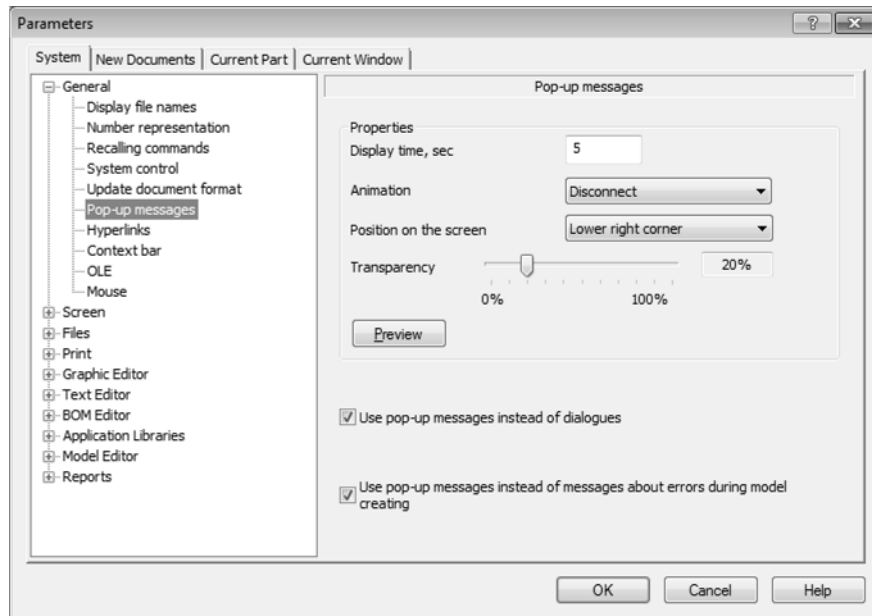


Fig. 9.1.1. Pop-up messages setup dialog

After closing a dialog with **OK** button, the information about system settings is stored in the file **.cfg* (see Section 9.5.2 on p. 1881) and used in the next working session.

Please pay attention to the following conventions and simplifications established in this Section:

- ▼ Further not all setup dialog, and only set control elements will be implied by “dialog” for adjustment of this or that object.
- ▼ Calling to this set of control elements will be described in the following form: **Service — Parameters... — System — Section Name** (group of setup objects) — **Subsection Name** (subgroup of setup objects). For instance, if the text says: “Setting of the pop-up message display parameters is done in the **Pop-up Messages** dialog. It appears on the screen after calling the **Services— Options... — System — General — Pop-up messages**. It means that you will have to perform these actions in order to access the pop-up message setup.

1. Select the item **Service** in the Main menu.
2. Select the command **Options...** in the list of commands of the menu **Service** that appears.
3. Activate the **System** tab in the displayed dialog.
4. In the object setup list expand the **General** section.
5. Select subsection **Pop-up Messages**.
The right part of the tab will display control elements with a general title **Pop-up Messages** to set parameters of messages.

9.1.2. General

9.1.2.1. File names display

Setting up the file name displayed in the window title and the number of files in the list of the most recently opened is done in the **General Settings** dialog. It appears on the screen after calling the **Services— Options... — System — General — File names displaying** command.

Tab. 9.1.1. Control elements of the file names setup dialog

Element	Setting description
Full File Path Name in Window Title Bar	Enable this option to have the path to the current document file and its name displayed in the system window title. When the option is disabled, the file path will not be displayed. The options of the file name display are shown in the field Example .
Remember the List of N Files	The field allows you to input or set with the counter the number of files in the list of the most recently opened. This list is displayed at the bottom of the File menu.

Upon finishing with settings, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.2.2. Number representation

Number display accuracy settings and angular units selection is done in the **Number Representation Setting** dialog. It appears on the screen after calling the **Services— Options... — System — General — Number representation** command.

Maximal accuracy for real numbers (coordinates of points, objects dimensions, variable values and so on)— 6digits.

Tab. 9.1.2. Control elements of the number representation setup dialog.

Element	Setting description
Numbers	The group serves to set up the display of real numbers display. This setting affects the fields and tables of the Property bar, the Values column of the Variables Window (see Section 7.1.1.2 on p. 1569) and the associated dimension value setting dialog (see Section 7.2.2.20 on p. 1614).
Number of Decimal Digits in Input/ Output Fields	The field allows you to input or set with the counter the number of decimal places for real numbers display.
Show Trailing Decimal Zeroes	The option allows to control padding with zeros the decimal positions of numbers if those have fewer decimal digits than prescribed.
Angular Units	The group serves to specify angular units. The selected units will be used to display angular values in the fields of the Property bar when creating and editing objects. When setting angular dimensions in graphic documents, the angles are measured in degrees, or in degrees and minutes, or in degrees, minutes and seconds, depending on the specified precision (see Section 9.2.6.10.12 on p. 1781).
Decimal System	Selecting this option means that angular values will be displayed in the $xx.xxx^\circ$ format with the specified precision.
Degrees, Minutes, Seconds	Selecting this option means that angular values will be displayed in the $xx^\circ xx' xx.xxx''$ format with the specified precision.
Radians	Selecting this option means that angular values will be displayed in the $xx.xxx\ rad$ format with the specified precision.

When finished with setting up the number scheme, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.



The measurement accuracy is customized directly in the measuring process.



Dimension value precision is defined by the parameters specified in the dimension text precision settings dialog (see Section 9.2.6.10.12 on p. 1781).



You can select desired angular units without calling the dialog. To do this, use the context menu by right clicking over empty space on any tab of the Property Bar.

9.1.2.3. Recalling commands

Enabling and disabling calling last commands from the context menu, as well as selection of the number of repeated commands, are made through the **Repeated command settings** dialog. It appears on the screen after calling the **Services — Options... command. — System — General — Recalling commands** command.

Tab. 9.1.3. Control elements of the recalling commands setup dialog

Element	Setting description
The “Redo” Command in the Contextual Menu	Enable this option so that the command to recent redo the last completed command is in the contextual menu. For instance, the Redo command will appear in the contextual menu after a line is created. Segment . Its invocation is equal to that of the Line command. If the option is disabled, the recent Redo Command command is not displayed in the contextual menu.
Remember the List of N commands	Enable this option so that the Recent Commands command is in the contextual menu. Its submenu contains a list of recent commands. Commands may be invoked in any sequence. To specify the number of stored commands, type or set with the counter a number— from 2 to 11. If the option is disabled, the list of commands is not displayed in the contextual menu.

Upon finishing with display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.2.4. System Management

Such working parameters of the KOMPAS-3D system as the access to a 3D modeling system and to a BOM design system are specified in the **System Management** dialog called by the **Service— Parameters... — System — General — System Management** command.

The dialog contains **Obtain KOMPAS-3D license** and **Allow working with BOM** options and the **Control of creation/recreation of associative views** group.

- ▼ The **Obtain KOMPAS-3D License** option allows to automatically obtain the license to work with a 3D design module when starting the KOMPAS-3D system

Licences are stored in the memory of the network HASP key. If there is a KOMPAS-3D license, then it will be obtained at the time the system is started. In this case, the **Service** menu **Obtain KOMPAS-3D License** command is marked with a “check”. It means that the license has been successfully obtained. If you need the license to be released during the working session, then call the specified command from the **Service** menu.

There are situations when licence obtaining is impossible. - **License Manager** is not responding (this is a special program that determines whether the user-requested license is available on the HASP key).

The KOMPAS-3D License may be impossible to obtain for the following reasons:

- ▼ - The key does not have the information about the product, whose license is being requested by the user (that is, there is no such license).
- ▼ the key has no free license available (i.e., all licenses are used by other users).

In case of failure to obtain a license, the work with 3D models is performed in Demo mode. In this mode, the words “demo mode” are displayed in the title of the respective-type document window of the corresponding type.

When working with models in the demo mode, the following actions are supported:

- ▼ View and modify the display type and orientation in the space
- ▼ Print
- ▼ Create a drawing using the **New Drawing from Model** command.

Editing a model and saving it on disk is not possible.

If during a working session there is an opportunity to acquire a license, then, to reserve it, call the respective command from the **Service** menu.

- ▼ The **Allow Work with BOM** option allows to automatically obtain the access to a BOM design system.

In this case, the **Service** menu **Allow Work with BOM** command is marked with a “check” that means the use of BOM design module. In this case, available are:

- ▼ All commands to work with BOM objects in BOM documents, graphic documents and model documents
- ▼ All commands to connect graphic documents and model documents to a BOM and the commands to synchronize a BOM with connected documents;
- ▼ Saving a BOM document.

If you need the BOM design module use to be cancelled during the working session, call the **Allow Work with BOM** command from the **Service** menu. After this, commands for working with BOM objects in graphical documents and model documents will become unavailable, and document synchronization in accordance with BOM settings will not be made. At the same time, creation of new document-BOM and objects in it is possible. The BOM document window will display the heading “Working with BOM is not allowed”; when attempting to save the document, a prompt will appear on the screen to enable working with BOM, since saving the *.spw file to disk is only possible if working with BOM is allowed.

- ▼ The **Control of creation/recreation of associative views** group is used for inclusion and setup of parallel creation and recreation of associative views.
 - ▼ If the **Allow parallel processing** option is enabled, several associative views may be created and recreated simultaneously. If the option is disabled, the views are processed one by one.
 - ▼ If the option **With lower priority** is enabled, view creation and recreation processes will be run with a lower priority. This feature makes it possible to keep working comfortably in other applications when processing views. However, if the system's workload is high, view creation and recreation processes may slow down. The **With lower priority** option is available, if parallel processing of views is enabled.
 - ▼ If the **Autosave before processing** option is enabled, documents are saved automatically before associative views are created or recreated. Auto saving is performed irre-

spective of the fact if it is enabled or disabled by default and only for those documents which have changed since the moment of last saving or auto saving. It is recommended to enable the **Autosave before processing** option, since auto saving is not performed during associative view processing.



Enabling parallel view processing makes sense if your computer has a multi-core processor. Maximal performance may be achieved, if your computer has 64-bit OS and sufficient RAM (more than 8 Gb).

When finished with setting up system parameters, click the **OK** dialog button. Note that enabling/disabling **Automatically obtain KOMPAS-3D license** and **Allow working with BOM** options becomes effective when the system is loaded for the next time.

To close the dialog without saving changes, press the **Cancel** button.

9.1.2.5. Document Layout Update

Text and graphical documents design update and BOM style update can be set up in the **Document design update** dialog. For more information about the layout update, see Section 10.2.1.2.1 on p. 1961.

It appears on the screen after calling the **Services— Options... — System — General — Document Layout Update** command.

Tab. 9.1.4. Control elements of the document layout update setting dialog

Element	Setting description
Do not update	Selecting this option means, that the automatic layout (style) update will not be provided. The document will be displayed with a layout (style) embedded in it. If necessary, you can manually update layout (see Section 10.2.1.2.2 on p. 1963) or style (see Section 6.2.7.1 on p. 1541).
Report about layout changing in the library	The option enabling messages about differences between the design (style) embedded in the document and the respective design (style) in the *.lyt library.
Update	Selecting this option means, that update of the layout (style) embedded to a document will be provided automatically if a difference between it and respective layout (style) in the *.lyt library is found. The message will popup upon completing an update.
Request Confirmation	The option enabling the confirmation request for the design (style) update prior to the update. You can confirm or reject the update. When using automatic update, it is recommended to enable the confirmation request to minimize the risk of occasional distortion in the design of documents created using the *.lyt library, which is different from the current library.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



The designs (styles) of already opened documents are updated in accordance with the setting that was applied while they were being opened. Therefore, to apply the layout update changes (for example, enabling of automatic update) it is necessary to close documents and reopen them.

9.1.2.6. Pop-up Messages

Setting of the pop-up message display parameters is done in the **Pop-up Messages** dialog. It appears on the screen after calling the **Services— Options... — System — General — Pop-up Messages**.

Tab. 9.1.5. Control elements of the pop-up messages setup options dialog

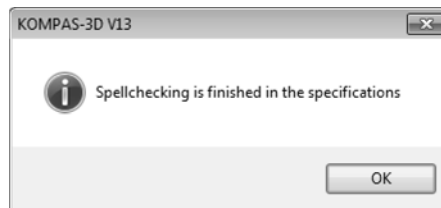
Element	Setting description
Properties	The group of elements allows to setup the mode of pop-up messages display.
Display time, sec	The field allows to specify the period of time pop-up messages are displayed for. Enter the value in seconds from 0,001 to 10000 on the keyboard.
Animation	The list allows to select the required visual effect for message appearance / disappearance. If No option is selected, then pop-up messages shall be displayed without any effects.
Position on the screen	The list allows to select the option of pop-up message location on the screen.
Transparency	The control defines the transparency of the pop-up message. <ul style="list-style-type: none"> ▼ If Transparency— 0% is set, then the image shall be completely non-transparent. To increase transparency, move the slider to the right. ▼ If Transparency— 100% is set, then the image shall be completely transparent. To change the transparency, use <▶> и <◀> key combinations, having selected the scale, e. g. with a the mouse click. The field located to the right from the scale is used to display the numerical value of the transparency.
Preview	The button allows to enable the demonstration of an example of pop-up message with specified properties before closing the dialog.

Tab. 9.1.5. Control elements of the pop-up messages setup options dialog

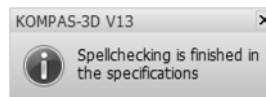
Element	Setting description
Use pop-up messages instead of dialogues	If this option is enabled, then pop-up messages will be displayed, if disabled— information dialogs (Fig. 9.1.2). Note that some help messages are pop-up message irrespective of this option's status. For example, the message about the number of points read from the file when creating a surface by the point grid.
Use pop-up messages instead of messages about errors during model creating	When the option is enabled, information about model construction errors will appear in the form of pop-up messages; if the option is disabled— in the form of dialogs (Fig.9.1.3).

Settings made in this dialog do not cover the pop-up **error messages**, which appear, for example, during the input of incorrect values in the dialog fields or the Properties toolbar fields (Fig.9.1.4).

When finished with setting up the pop-up messages, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

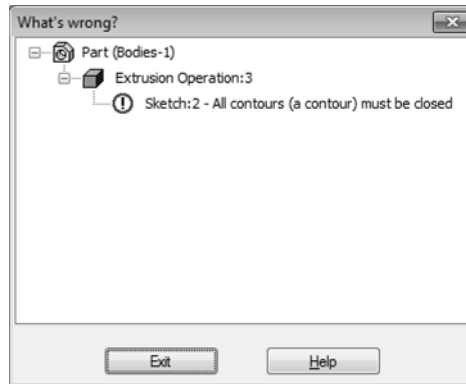


a)

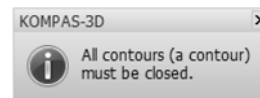


b)

Fig. 9.1.2. Examples of information message display:
a) dialog form, b) pop-up form



a)



b)

Fig. 9.1.3. Examples of information message display
Error in Construction of Model:
a) dialog form, b) pop-up form

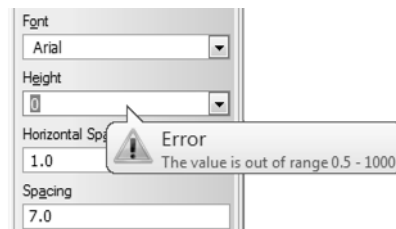


Fig. 9.1.4. Example of error message during parameters input


9.1.2.7. Hyperlinks

Setup of hyperlink parameters is performed using the **Hyperlink** dialog which is called by the **Service — Parameters... — System — General — Hyperlinks**.

Tab. 9.1.6. Control elements of the Hyperlinks Setting dialog

Element	Setting description
Automatic hyperlinks creation	If the option is enabled, while creating cut/section line, view direction arrows and enlarged detail, hyperlinks from the designation to automatically created view will be formed (see Section 3.3.14.1 on p. 1110). If the option is disabled, hyperlinks are not created automatically.

Tab. 9.1.6. Control elements of the Hyperlinks Setting dialog

Element	Setting description
ALT + click to Follow Hyperlink	If the option is enabled, you should click the object while holding down the <Alt> key to activate the hyperlink. If the option is disabled, hyperlink is activated just by clicking the object.
Display Hyperlinks in Pop-up Tips	If the option is enabled, the name of target object will be displayed in the pop-up tip (see Fig.11.2.1 on p. 2018) as the cursor approaches the hyperlink object. If the option is disabled, the name of the target object is not displayed in the pop-up tip.
 Display hyperlink icon on the cursor	If the option is enabled, a hyperlink icon is displayed near the cursor as the cursor approaches the hyperlink object. If the option is disabled, the icon is not displayed near the cursor.

Having finished the hyperlink setup, press the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.2.8. Context bar

You can set up context bar display parameters in the **Context Toolbar** dialog that is called by the following command: **Service— Parameters... — System — General — Context bar**.

Tab. 9.1.7. Control elements of the context bar parameter setup dialog

Element	Setting description
Use context bar	The group of options that enables or disables the context bar display when working with documents of different types.

Tab. 9.1.7. Control elements of the context bar parameter setup dialog

Element	Setting description
Display context bar	<p>The group of options that defines how the context bar will be displayed. This setting is used for all types of documents where the context bar use is enabled.</p> <ul style="list-style-type: none"> ▼ If both options are disabled no context bar will appear. ▼ If both options are enabled, the context bar shall be displayed when objects are selected or the context menu is called. ▼ If at selecting option is enabled and at context menu call option is disabled, then the context bar shall be displayed when an object is selected and disappear after the context menu of the selected object is called. ▼ If at context menu call option is enabled and at selecting option is disabled, the context bar shall be displayed only after calling a context menu with some objects being selected[*]. If context menu is called during some command execution (for example, during circle construction), then regardless of the setting context bar does not appear.

* Context bar shall be also displayed in a text document when a context menu is called without any objects being selected.



When selecting some objects (for example, OLE-insertions), the context bar will not appear.

Having finished the context bar setup, press the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.2.9. OLE

Format selection for the file to be used when inserting KOMPAS documents into other documents using OLE technology (see Section 12.2 on p. 2051), is made in the **OLE** dialog called by the following command: **Service – Parameters.... – System – General – Hyperlinks**.

The dialog has two options: **EMF** and **BMP**. By default, ELF format is used for OLE-inserts. This reduces the size of the file containing insertions.

However, in the documents of some applications (for example, Open Office pack applications), this format does not ensure correct processing of OLE-insertions. In these cases it is recommended to use BMP format.

Upon finishing the settings, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.2.10. Mouse

You can reverse the direction of the mouse's wheel rotation that is used for zooming in/zooming out the image.

The mouse wheel rotation is set up in the dialog which is called by the **Service — Parameters... — System — General — Mouse** command.

The dialog contains the **Reverse direction of the mouse wheel rotation** option.

If the option is disabled, rotation of the mouse wheel in the direction to the user zooms the image out, while rotation of the wheel off the user zooms the image in. If the option is enabled, the result of the mouse wheel rotation will be the opposite.

Upon finishing the settings, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.3. Screen

9.1.3.1. Working Field Background

Background color setup for graphic, text documents and BOMs is done in the **Background Color Settings** dialog. It appears on the screen after calling the **Services— Options... — System — Screen — Working field background**.

In this dialog you can perform the following actions.

- ▼ Set the document background color using the **Color of Working Field Background** group of elements.
- ▼ Set the background color for editing text using the **Text Editing Background Color** group of elements.

The text editing background color is the color used as the background when creating and editing the following objects:

- ▼ Drawing annotation text and tables
- ▼ Labels as part of designations
- ▼ Tables in a text document,
- ▼ BOM items.

Tab. 9.1.8. Control elements of the background color setup dialog.

Element	Setting description
Window Color Set by Windows	To make the background color match the overall Windows settings, enable this option. To select the working field background color that differs from Windows settings, disable the option. After disabling the option, a button for selecting color becomes available.

Tab. 9.1.8. Control elements of the background color setup dialog.

Element	Setting description
Color	The button allows to choose background color in the standard color selection dialog. It is accessible when the option Window Color Set by Windows is disabled.
Color Preview Window	The window serves to display the color set with the help of Color button.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



Document color displaying setup does not cover preview mode (see Section 8.2 on p. 1639). Background color in this mode— white, and object colors depend on the output settings.

9.1.3.2. Background of Model's Working Field

Working Field Background Color setting for model documents is performed in the **Model Background Color Settings** called with the **Service — Parameters... — System— Screen— Background of Model's Working Field** command. In this dialog you can set up solid background color or background with gradient transition from one color to another one.

Tab. 9.1.9. Control elements of the model background color setup dialog.

Element	Setting description
Window Color Set by Windows	To make the background color match the overall Windows settings, enable this option. To select the working field background color that differs from Windows settings, disable the option. After disabling the option, a button for selecting color becomes available.
Color	The button allows to choose background color in the standard color selection dialog. It is accessible when the option Window Color Set by Windows is disabled. Identical buttons are located in the group of gradient transition color setup. Available when the Use Gradient Transition option is enabled.
Use Gradient Transition	The option serves to use gradient transition between two colors when setting up background color. When working with 3D models, the top and bottom portions of the document field is set to the selected colors. A smooth transition is established from one color to another one between them. After option enabling, buttons of color setup become available.
Top color Bottom color	Those are the windows of colors, between which the gradient transition is made. Each of colors is setting up using its respective Color button.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



Document color displaying setup does not cover preview mode (see Section 8.2 on p. 1639). Background color in this mode— white, and object colors depend on the output settings.

9.1.3.3. Text background

Text's background is set up in the **Text background** dialog. It appears on the screen after calling the **Services— Options... — System — Screen — Text background** command. In the dialog, you can set up the color and transparency of the text background near the cursor.

Tab. 9.1.10. Control elements of text background setup dialog

Element	Setting description
Background color	After enabling the option, a button for selecting color of the text background becomes available. If the option is disabled, text near the cursor will have no background.
Color...	The button allows to choose background color in the standard color selection dialog. It is available if the Background color option is enabled.
Transparency	A scale for adjusting the text background transparency. In order to change the background transparency, use the mouse to move the slider: to the right to increase the transparency level, or to the left to decrease it. The scale is available if the Background color option is enabled.

Having finished adjustment of text background, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.3.4. Color Scheme

Setting up colors for displaying document and system elements is done in the **Color Scheme Settings** dialog called by the **Service— Parameters... — System— Screen— Color Scheme** command.

Document items are the items making up the document contents. These elements are created by the user: curves, points, hatches, dimensions, designations in a graphic document; bodies and surfaces in the document-model.

The system items are various auxiliary items that can be displayed in a document window (grid, phantoms, cursor tips, etc.). Additionally, the color set up for system elements is used to display the following objects:

- ▼ Selected and highlighted document elements
- ▼ Large cursor
- ▼ Background views and layers

- ▼ Frames of hidden and associative views
 - ▼ Marquee frames displayed on screen when selecting objects and when zooming in by marquee
 - ▼ Designations of coordinate system in a graphic document and sketch,
 - ▼ Outline frames for bitmap and OLE-objects, etc.
- Options available when setting up element colors are shown in the Table9.1.11.

Tab. 9.1.11. Item color setting options

Variant	Setting description
Specified during setup	Elements are displayed in the same colors that were specified when setting them up or at creation.
Inverse to Background Color*	If an element own color (defined when it was set up) coincides with the background color or is very close to it, then the element is displayed in the color which is the inverse of the background color.
Define	All elements and objects are displayed in one color. The colors defined during their setup will be ignored. After activating the option Define , the Color button becomes accessible; it serves to select the color of elements becomes accessible.

* If a background fill with gradient transition is set for models, then the inverse colors of elements in models documents are not supported. The 3D features and system elements are displayed in the colors defined at their setup.

When finished with setting up the Color scheme, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.



Document color displaying setup does not cover preview mode (see Section 8.2 on p. 1639). Background color in this mode— white, and object colors depend on the output settings.

9.1.3.5. Color of text elements

Customizing the color of text elements that are displayed in the document window is performed in the **Color of Text Elements** dialog. It appears on the screen after calling the **Services—Options... — System — Screen — Color of text elements**. In this dialog you can specify the font color of text elements and the underlining color for spell checking.

Tab. 9.1.12. Controls of the background color of text elements setup dialog.

Element	Setting description
Color...	The button allows to specify the color of the text element being adjusted. After pressing the key, the standard Windows color selection dialog will be displayed.

Tab. 9.1.12. Controls of the background color of text elements setup dialog.

Element	Setting description
Links	<p>The group controls make it possible to select the color of links with different statuses. The Color... button is available, if the option related to the link's status is enabled. If the option is disabled, it is impossible to adjust color and the link will be displayed in black color.</p> <ul style="list-style-type: none"> ▼ Valid links are the links that retain relation to the source object. ▼ Invalid links are the links that have lost relation to the source object (see Section 4.1.4.3 on p. 1337 for more information about links to objects). ▼ Links that require update are links in which changes in the source are not yet reflected. Links that require update can appear in reports with data in the form of links (see Section 5.2.3.7 on p. 1432) and in associative reports (see Section 5.2.3.8 on p. 1433) after modification of the document which is the source of the report.
Letter symbols being sorted	<p>The option enables to manage drawing color layout of the symbols for which the Auto-sorting is enabled (see Section 3.3.12.20 on p. 1067). When the option is enabled, the Color... button that allows to set up color layout of the symbols is available. If the option is disabled, the Color... button is unavailable. Symbols will be displayed in black.</p>
Reference numbers with BOM items	<p>The option allows to control the color layout of reference items that are included to BOM items. When the option is enabled, the Color... button that allows to set up color layout of the reference items is available. If the option is disabled, the Color... button is unavailable. Symbols will be displayed in black.</p>
Underline	<p>The controls of this group allows to define the color of underlining of the following text fragments:</p> <ul style="list-style-type: none"> ▼ those supposed to contain spelling and grammar errors following a spell-check, ▼ those for which no spelling check language is specified or the <i>Do not check</i> option is selected as the language attribute. <p>On spell check, see Section 4.1.3 on p. 1321.</p>

Having finished the text color setup, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



Document color displaying setup does not cover preview mode (see Section 8.2 on p. 1639). Background color in this mode— white, and object colors depend on the output settings.

9.1.3.6. Properties Bar

Properties bar displaying parameters customizing is performed in the **Property Bar Appearance** dialog. It appears on the screen after calling the **Services— Options... — System— Screen— Property bar** command.

Tab. 9.1.13. Control elements of the Properties bar layout setup dialog

Element	Setting description
Page Tabs	Group of elements makes it possible to set up the appearance of the page tables and titles on them.
Font	The drop-down list allows you to select the font type. The list contains all fonts enabled in Windows.
Color	The button allows to change the symbols color of active tab name. It calls the standard Windows color selection dialog. The names of inactive tabs are displayed in the black.
Size in pixels	The field serves to set up symbol height value (in pixels). The value is entered manually or specified using the counter.
Italic	The option allows to set the italic style of characters of the active tab title.
Bold	The option allows to set the bold style of characters of the active tab title.
View	<p>Display options of Page Tab Tags.</p> <p>When selecting the 3D option:</p> <ul style="list-style-type: none"> ▼ the icons indicating the purpose of their respective tabs will be displayed on the tab tags along with the labels, ▼ Tab tag width will be changing according to the width of the Property bar, so one does not need to scroll through the tags. <p>When selecting the Flat option, only the labels will be shown on the tags, with the tags width fixed.</p>
Parameters	The group of elements makes it possible to customize display of the parameters' names and position tolerance control buttons.
Show names	<p>Using this option means that the names will be displayed next to interface items.</p> <p>If this option is active, you can select the method of displaying parameter names— Optimum or Always.</p>
Tolerance control buttons	If this option is enabled, it means that near the value (linear, angular) field on the Property bar, there will also be the button for controlling the value's tolerance.
Reset	The button serves to restore default Properties bar displaying parameters.

Tab. 9.1.13. Control elements of the Properties bar layout setup dialog

Element	Setting description
Preview Window	The window serves to display the appearance of the objects to be set up. All made changes are immediately shown in this window.

Upon finishing with Property bar display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

If the Properties Bar is on the screen, the selected dialog can be called from its context menu using **Property Bar Appearance** command.

In addition, you can customize appearance of some Properties bar elements using the **Show Settings Names** and also **Tab Tags View** context menu commands.



You can also customize icon size on the Properties Bar, see Section 9.1.3.8.7 on p. 1690.

9.1.3.7. Document tabs

Display parameters for document tabs and their appearance are set up in the **Setup of document tabs** dialog. It appears on the screen after calling the **Services— Options... — System — Screen — Document tabs** command. You can also call the dialog using the **Setup...** command from the context menu of the document's tab.

Tab. 9.1.14. Control elements of the document tab setup dialog.

Element	Setting description
Show tabs	The group of options that controls displaying tabs on the screen. If the Show tabs option is enabled, tabs of opened documents are displayed on the screen.
At the top	If the option is enabled, document tabs are located above the current document's window.
At the bottom	If the option is enabled, document tabs are located beneath the current document's window.
Full file name	If the option is enabled, the name that includes full path to the file is displayed in the tab.
Show icons	If the option is enabled, the icon of the document type is displayed in the left part of the tab.
&Colored document tabs	If the option is enabled, tabs are displayed in different colors.

Tab. 9.1.14. Control elements of the document tab setup dialog.

Element	Setting description
Show the button for quick closing of the active tab	<p>If the option is enabled, the Close button for closing the active window is displayed in the right part of the active tab.</p> <p>If the option is disabled, the Close button is displayed in the right part of the tabs.</p>
Tab width	The group of elements that controls the width of tabs.
Auto setting	Choosing this option means that all tabs are displayed in the KOMPAS-3D window. If the tab's width is less than the length of the file's name, the file's name will be trimmed and ended with ellipsis.
Maximum tab width, pixels	<p>Choosing this variant means that the maximum width of a tab cannot be more than the prescribed value (in pixels). Tabs are displayed with different width depending on the file name's length. If the tab's width is less than the length of the file's name, the file's name will be trimmed and ended with ellipsis.</p> <p>If all tabs do not fit in the KOMPAS-3D window, one or several tabs will not be displayed.</p> <p>The available range of the tabs' width is 50-1000 pixels.</p>
By file name	<p>Selecting this option means that the file's full name is displayed on the tab. Tabs are displayed with different width depending on the file name's length. If all tabs do not fit in the KOMPAS-3D window, one or several tabs will not be displayed. If the tab's width exceeds the KOMPAS-3D window, the file's name will be trimmed and ended with ellipsis.</p>

When finished with setting up document tabs display, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.3.8. Interface setup

9.1.3.8.1. Commands

You can customize Main Menu and toolbar contents (see Section 9.4.2.1 on p. 1872).

This setup is performed in the **Commands** dialog which appears after calling one of the following commands: **Service — Parameters...** — **System — Screen — Interface setup — Commands** or **Service — Interface setup — Commands**.

Tab. 9.1.15. Control elements of the command contents setup dialog

Element	Setting description
Categories	The field contains the list of command categories in the KOMPAS-3D system.

Tab. 9.1.15. Control elements of the command contents setup dialog

Element	Setting description
Commands	The field contains the list of commands belonging to the selected category.
Description	The field provides the description of the selected command purpose.

Choose command category and name in the corresponding fields. “Drag” command to the desired bar or menu.

Click the **OK** or **Cancel** buttons to close the dialog.

It shall be noted that the changes made will not be canceled when clicking the **Cancel** button. To return the interface to the previous view the corresponding setting shall be changed.

9.1.3.8.2. Toolbars

You can create user-defined toolbars and place on them any commands (see Section 9.4.2.2 on p. 1874). For this purpose, the **Tool bars** dialog is used which is called by one of the following commands: **Service — Parameters... — System — Screen — Interface setup — Tool bars** or **Service — Interface setup — Tool bars**.

Tab. 9.1.16. Control elements of toolbar setup

Element	Setting description
Panels	The field contains the list of toolbars in the KOMPAS-3D system. A “check” next to the panel name means the panel is currently displayed in the program window. To hide a toolbar, click the mouse on the “check” by its name.
Reset	This button serves resetting the selected toolbar to the default state. If you added or deleted toolbar commands or manipulated with its display, then after clicking the Reset button all changes shall be undone.
Reset All	This button serves to reset all toolbars and menus to the default state.
New...	This button serves to create a user toolbar.
Rename	This button serves to rename a user toolbar.
Delete	This button serves to delete a user toolbar.
Labels for Buttons	Activate this option to have command names displayed next to the respective buttons on the selected toolbar*.

* If a bar is docked by the top border of the window, the labels cannot be displayed.

Click the **OK** or **Cancel** buttons to close the dialog.

It shall be noted that the changes made will not be canceled when clicking the **Cancel** button. To return the interface to the previous view the corresponding setting shall be changed.

9.1.3.8.3. Utilities

You can add user commands for opening application files, documents etc. to the KOMPAS-3D system menu **Service** (see Section 9.4.2.4 on p. 1876).

Command adding is performed in the **Utilities** dialog. It appears on the screen after calling the **Services— Options... — System — Screen — Interface setup — Utilities** or **Service — Interface setup — Utilities**.

Tab. 9.1.17. Elements of **Utilities** dialog control

Element	Setting description
Menu	<p>This is a list of user commands for calling utilities. The control buttons are located above the list.</p> <p>The New button serves to add a new command to the list. After clicking this button, an empty line appears at the end of the list. Enter the command name there.</p> <p>The Delete button serves to delete the selected command from the list. The buttons Move Up and Move Down manage the location of commands in menu.</p>
Command	<p>The field contains the full name of the file to be opened when the command is called. It may be entered from keyboard though it is recommended to use the standard controls of the operation system. To do this, use the Browse button located to the right from the field. The standard Windows Open File dialog will appear on the screen. Select the desired file and press the OK button. The full name of the file shall appear in the Command field. If an executable file is selected, then the created command shall launch the application. If a non-executable file is selected, then the created command shall launch the Windows application associated to that file type.</p>
Parameters	Field for specifying the command running parameters.
Working directory	The field to specify the path to the directory of application that is launched by the command.

Click the **OK** or **Cancel** buttons to close the dialog.

It shall be noted that the changes made will not be canceled when clicking the **Cancel** button. To return the **Service** menu to the previous view the corresponding setting will be changed.

9.1.3.8.4. Keyboard

You can assign keys or key combinations for quick calling the commands (see Section 9.4.2.3 on p. 1875).

This setup is performed in the **Keyboard** dialog which appears after calling one of the following commands: **Service — Parameters... — System — Screen — Interface setup — Keyboard** or **Service — Interface setup — Keyboard**.

Tab. 9.1.18. Control elements of the keyboard shortcut settings dialog

Element	Setting description
Categories	The field contains the list of command categories in the KOMPAS-3D system.
Commands	The field contains the list of commands belonging to the selected category.
Description	The field provides the description of the selected command purpose.
Assign Shortcut	The drop-down list contains the list of KOMPAS-3D system document types. You can select the document type for which the key combination will work to call the command. Selecting the By default option means the key combination will be valid when working with document types in which it is not reserved for the system and is not used to call another command. When working with documents of other types, the setting is ignored and the combination is used according to its purpose.
Current	The field contains the list of active key shortcuts to call the selected command.
New Keyboard Shortcut:	The field to specify key or key combination assigned to command. To define a key or a shortcut, simply press the desired key(s) on the keyboard. The New Keyboard Shortcut field will display the name(s) of the pressed key(s), and an info message will appear below it. If the selected key(s) is already in use for calling another command, its name will be displayed (if the command is not contained in any menu and called, for example, by a button on the Special Control bar, the Command outside categories line is displayed); if the button is not in use, the Not related line is displayed.
Assign	The button allows to assign defined key or key combination to a command. Available if this key or key combination is not assigned to other command.
Delete	The button allows to delete key(s) name from the list of those assigned to the given command.
Reset All	This button makes it possible to restore the default key combinations for calling KOMPAS-3D system commands.

Click the **OK** or **Cancel** buttons to close the dialog.

It shall be noted that the changes made will not be canceled when clicking the **Cancel** button. To return to the previous assigns the setting shall be changed again.

9.1.3.8.5. Menu

The control of the display of the Main Menu and context menus of window tabs is done in the **Menu** dialog. It appears on the screen after calling the **Services— Options... — System — Screen — Interface setup — Menu** or **Service — Interface setup — Menu**.

Tab. 9.1.19. Control elements of the menu displaying setup dialog

Element	Setting description
Show Menu	From the drop down list, select a variant of the Main Menu which is required to display in the current setup window. To change the menu contents, switch to the Commands tab.
Show context menu	To setup the context menu of window tabs, select the Window Tabs line in this list. To change the menu contents, switch to the Commands tab.
Reset	This button cancels all changes in the menu (Main or context), i.e. sets up its default appearance.
Menu animations	Drop down list of visual effects which may accompany the opening of the command list.
Menu Shadows	The option serves to show shadows when displaying menu.

Click the **OK** or **Cancel** buttons to close the dialog.

It shall be noted that the changes made will not be canceled when clicking the **Cancel** button. To return the interface to the previous view the corresponding setting shall be changed.

9.1.3.8.6. Parameters

You can set up display parameters:

- ▼ pop-up tips i. e. tags on toolbar and compact panel buttons,
- ▼ toolbar and compact panel buttons,
- ▼ Main Menu commands.

This setting is done in the **Parameters** dialog. It appears on the screen after calling the **Services— Options... — System — Screen — Interface setup — Parameters** or **Service — Interface setup — Parameters**.

Tab. 9.1.20. Control elements of the interface element setup dialog.

Element	Setting description
Show Screen Tips on toolbars	The option allows to control the display of pop-up tips with command names when the cursor is over a button.
Include key combinations in Tool Tips	The option allows to include key combinations for fast command call to pop-up tips. The option is available when the display of pop-up tips for buttons is enabled.

Tab. 9.1.20. Control elements of the interface element setup dialog.

Element	Setting description
Menus show recently used commands first	The option allows to show the most used menu commands first. If disabled, all commands will be displayed simultaneously.
Show full menus after a short delay	The option controls the display mode for full menu. If you delay the mouse cursor over an open menu whilst this option is enabled, then after a short delay the menu will display all of its commands. When disabled, you will need to press the Expand button (the last line in a partially opened menu) to view the full menu.
Reset my usage data	The button allows to restore the default state of a list of commands displayed first in a menu being opened when the option Menus show recently used commands first is enabled.

Click the **OK** or **Cancel** buttons to close the dialog.

It shall be noted that the changes made will not be canceled when clicking the **Cancel** button. To return the interface to the previous view the corresponding setting shall be changed.

9.1.3.8.7. Icon size

Menu commands and command calling buttons icon size can be changed in the toolbars. For this purpose, the **Icon size** dialog is used which is called by one of the following commands: **Service — Parameters... — System — Screen — Interface setup — Icon size** or **Service — Interface setup — Icon size**.

Tab. 9.1.21. Control elements of icon size setting dialog

Element	Setting description
Single	The drop-down list allows to set up size of the control elements (switches, buttons, icons, etc.) for all interface elements. This list is available by default— its respective option is enabled. To customize size of controls separately from every interface element, disable the option.
Toolbars	The drop down list allows to set the size of command buttons and their icons.
Menu	The drop down list allows to set the size of icons for menu commands in KOMPAS-3D system.
Property Bar, Properties and Numbering windows	

Tab. 9.1.21. Control elements of icon size setting dialog

Element	Setting description
Controls	The dropdown list makes it possible to set the size of control elements on the Property bar tabs (switches, options, and so on) and element size in table for Properties and Numbering windows.
Advanced Control Bar	You can use the dropdown list to set the size of buttons in the Special Control Panel of the Property bar and the size of buttons in the toolbar for Properties and Numbering windows .
Document Tree	You can use the dropdown list to set the icon size in the Document tree, as well as in the tree of the Property editor window's tree.



Menu commands and toolbar buttons can be displayed as icons, text or icons with text. Selection of display option is described in Section 9.4.2 on p. 1872.

Upon finishing with settings, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.4. Files

9.1.4.1. Allocation

At any time while working with the system, you can get information about location of *KOMPAS.ini* file, configuration files, folders for temporary and system files, folders for document files, libraries, templates, profiles. For this purpose, the **System file location** dialog is used which is called by the following command: **Service — Parameters... — System — Files — Location**. The current paths to the above listed files and folders are displayed in this dialog.

The list of files and folders is shown in Table9.1.22.

Tab. 9.1.22.

Name	Description
Kompas.ini file	The <i>KOMPAS.ini</i> file contains information about the location of configuration files and system folders. The <i>KOMPAS.ini</i> file is used if the location should be different from the default one (the information about the default location of those files and folders is stored within the system in the form of KOMPAS-3D environment variables KOMPAS-3D, environment variables are described in Section 9.5.4 on p. 1886). Besides the mentioned data, it may contain additional information about system running operation parameters.
New Document Parameters	The file <i>*.prj</i> contains information on the settings of new documents i. e. on all settings made on the New Documents tab of the general setup dialog.
System Parameters	The file <i>*.cfg</i> contains information on system settings i. e. on all settings made on the System tab of the general setup dialog.
Working Window Parameters	The file <i>*.dsk</i> contains information on the system working window settings.
Temporary Files	Indicates the folder to hold temporary files, as well as document files which were automatically saved during a KOMPAS-3D session.
Working directory	This is the folder to place KOMPAS-documents. This folder will be offered by default on the first time you open or save a document.
System Files	The folder containing system files: user menu files, allowances, material density reference file, style libraries and so on, When setting up text templates, libraries of holes and sketches, then by default, this folder is suggested for placing <i>graphic.tdp</i> , <i>holelib.lfr</i> , and <i>skchlib.lfr</i> files.
Library Files	The folder containing library files. By default, this folder will be offered first when adding libraries to the Library Manager.
Templates	The folder containing KOMPAS document template files. When creating a document, the templates located in this folder shall be offered. The same folder is offered by default to hold all newly created templates.
User Profiles	The folder containing profile files. When selecting profiles, those located in this folder will be shown. The same folder is offered by default to hold all newly created profiles.
Bend Tables	The folder containing bend table files. When selecting bend tables, the tables located in this folder will be shown. The same folder is offered by default when saving a bend table from a sheet model into a file.

Tab. 9.1.22.

Name	Description
Thread tables	The folder containing thread table files. When selecting thread tables, the tables located in this folder will be shown.
Templates of multilines	This is the folder containing the templates of multilines. When creating a document, the templates located in this folder shall be offered. The same folder is offered by default to hold all newly created templates.
Plotter/printer configuration	The folder containing printing devices configuration files. At saving of the configurations, this folder will be suggested as default. When loading configurations, those located in this folder will be offered.
Image files	The folder containing the files with image examples that can be used for checking the part surface smoothness by the Environment reflection method.
Collection of mathematical expressions	The <i>collection.law</i> file containing mathematical expressions that may be used for assigning values to variables in the Variables window and setting point coordinates when creating a curve by law. Expressions contained in the file are displayed in the Insertion of mathematical expression dialog when choosing the From collection file group .
File of unspecified maximum dimensional deviations	The <i>Gen_tol.ttl</i> file that contains values of maximum deviations of linear and angular dimensions by general tolerances. These values are used in models for dimensions of sketches and parameters of operations to which no individual tolerances are assigned.

If the KOMPAS-3D system was started without using the *kViewer.ini* *KOMPAS.ini* file, then the dialog displays the default path to files and folders (the path to the *kViewer.ini* *KOMPAS.ini* file is missing). To write the file *KOMPAS.ini*, click the **Create Kompas button**. The file *KOMPAS.ini* containing the current settings will open in the application registered in Windows for ini-type files. If there are no registered applications, select any text editor in the dialog that appears. Edit and save the file. Structure and syntax of the file *KOMPAS.ini* is described in Section 9.5.3 on p. 1883.



When the system KOMPAS-3D is started, search for the *KOMPAS.ini* file is only conducted in the folder with *KOMPAS.exe*. This is why the newly created *KOMPAS.ini* is automatically placed in this folder. Placing it to any other folder has the same effect as if it does not exist.

If *KOMPAS.ini* file is already presenting at the time the system is started, then the dialog displays the current path to this file. To edit the file, click the *kViewer.iniKompas.ini* button. Make the required changes and save the file.

A new newly created or edited *KOMPAS.ini* file will be used in the next KOMPAS-3D working session.

You cannot undo a creation or an editing of the *KOMPAS.ini* file. If required, delete (rename) the file on the disk or edit it again.

9.1.4.2. Set Access Rights

When organizing joint network operation, you can set parameters of access to subordinate files (style libraries, property types, designs and so on), as well as enable control over the subordinate files changing and changing the source files for insertions made by an external link (fragments, components and so on) and documents that have been opened for reading.

You can set access rights for files in the **Setting access rights** dialog which is called by the **Service — Parameters... — System— General— Setting access rights** command.

Tab. 9.1.23. Control Elements of Access Rights Setup dialog

Element	Setting description
Access Level to Linked Files	A dropdown list allows to set up access right to linked files (style libraries, attribute types, etc.). Linked files in relation to documents are such files as libraries of styles, layouts, types of attributes etc. Access Level options are shown in Table 9.1.24.
File Modification Control	This option makes it possible to control changes of jointly used files, as well as files that have been opened for reading only. By default, the option is enabled. Activating this option means that the system will be automatically checking whether the file has been modified by any of the other users during your working session. In case of a positive check verification result, the message with invitation to reread file will appear on the screen (see Section 11.3.1.3 on p. 2027).
Interval check, min	The field allows you to type or set with the increment button a time interval after which the system will monitor change of modifications to shared files. By default the interval is equal to 1 minute.
Notify When Only Files Opened Are Read-Only Ones*	The option allows you to control notifications when you open read-only files. The option is available if the file changing control is activated. Notifications are deactivated by default.
Display access for component Read only*	You can use this option to display the Read only icon in the model's Design Tree or components (parts, assemblies), the source files of which are available for viewing only (see Appendix I). This icon is displayed before the <i>Components</i> section, group or branch of identical components, if all source files are available for viewing only. By default, the displaying is enabled.

* In this case, the **Read only** access means that the file was assigned the “Read only” system attribute in the file’s properties.

The option does not relate to assigning *temporary read only* access to a file that is being opened, for example, by enabling the “Read only” option in the file open dialog.

Tab. 9.1.24. Access levels to linked files

Access Level	Description
Enable Reading	Makes it possible to read only the files at other workplaces that are subordinate to documents opened at the workplace. For example, if you opened a drawing using line style from some library, then other users (on other working places) will be able to apply styles from this library, but not to edit them. Besides that, they can copy the file of the line style library that you are using but cannot change or delete it. Once the drawing that uses the library has been closed, it will become fully available for other users.
Enable Reading and Saving	Makes it possible to read only the files at other workplaces that are subordinate to documents opened at the workplace. If File Modification Management option is enabled, then in case of linked file changing by other user, you will get message about it (see Section 11.3.1.3 on p. 2027).

Upon finishing the file access permissions settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.4.3. Backup copy

Backup copy provides maximum safety of the documents. So it is recommended to always use at least one option of backup copying files.

The backup is set up in the **Backup document copying** dialog which is called by the **Service – Parameters... – System – General – Backup copying** command.

Tab. 9.1.25. Control elements of the backup copy setting dialog

Element	Setting description
Save Original Copy	If this option is enabled, the system saves the source copy of the document in the specified folder. The copy is saved before executing the first command of the document saving on disk during the working session. Saving includes copying the full access path to the file. For example, if we are working with a drawing saved to the file <i>c:\Work\First.cdw</i> , while having the <i>d:\Original</i> folder specified to save original files, then the copy of the original file will be saved as <i>d:\Original\c\Work\First.cdw</i> .
Browse...	The button serves to specify folder for original document copy saving. The button is available when the Save Original Copy option is enabled. After folder selection, the full path to it is displayed in the To Folder field.
Save Previous Copy	If this option is enabled, and a folder for saving is specified, the system saves the previous copy of the document. The copy is saved each time the first command of the document saving on disk is executed during the working session. Saving includes copying the full path to the file. For example, if you are working with the drawing <i>c:\Work\First.cdw</i> , and the folder for previous copies is <i>e:\Backup</i> , then the previous copy will be saved as <i>e:\Backup\c\Work\First.cdw</i> . If the option is enabled, but no folder name is specified, then the previous copy will be saved to the same folder as the document itself. The copy's file name is formed in accordance with the following pattern: <document's file name> + <.> + <name extension> + <.> + <bak> (for example, <i>example.frw.bak</i>). Restoring documents from *.bak files is described in Section 11.3.2.2 on p. 2029.
Browse...	The button serves to specify folder for previous document copy saving. The button is available when the Save Previous Copy option is enabled. After folder selection, the full path to it is displayed in the To Folder field.
In one directory with Document	The button allows to cancel previous copy saving in folder specified using Browse... button. After pressing the In One Directory with Document button, the To Folder field will be cleared. The copy will be saved to the same folder and with the same name as the document, but with the extension *.bak.

Tab. 9.1.25. Control elements of the backup copy setting dialog

Element	Setting description
Save "mirror" Copy	<p>"Mirror" copying is an additional method of backing up data, when an additional duplicate document is created in another location on the same disk or on another physical disk (for example, on an enhanced-reliability remote server disk).</p> <p>If the option is enabled, then the system will be saving a "mirror" copy of the document to the specified folder each time when the save to disk command is executed during a working session. Saving includes copying the full access path to the file. For example, if you are working with the drawing <i>c:\Work\First.cdw</i>, and the folder for previous copies is <i>g:\Mirror</i>, then the mirror copy will be saved as <i>g:\Mirror\c\Work\First.cdw</i>.</p>
Browse...	<p>The button serves to specify folder for "mirror" document copy saving. The button is available when the Save Mirror Copy option is enabled. After folder selection, the full path to it is displayed in the To Folder field.</p>

Upon finishing with backup copy settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

In the folders specified for backup copies, the structure of files is created during file saving that corresponds to the location of the main document file on the disk. The name of the upper folder in the structure to be created corresponds to the disk name where the main file is being saved. For example, drawing *First.cdw* is saved in *C:\Work* folder, while having the *D:\Reserve* folder specified to save copies. The full path to the document copy file will be the following *D:\Reserve\C\Work\First.cdw*.

It is not recommended to select similar folders for saving reserve copies of different types, because in this case it is possible to lose information. If you attempt to finish backup copy setting up with the same parameters, the warning message will appear on the screen (Fig.9.1.5).



Fig. 9.1.5.

This message buttons allow to confirm setup or cancel it. In case the settings is canceled, you can change folders for backup copying. If setup is confirmed, backup copy will be performed. Meanwhile, only one file is created which is the same as the current state of the saved main file.

9.1.4.4. Auto Save

Automatic saving of documents ensures maximum safety of working results in case of any failures (emergency exit from the system, electrical power interruptions, PC hardware faults and so on). Therefore, while working with system it is recommended to always enable auto save.

You can set up auto save parameters in the **Auto Save Document Files** dialog that is called by the following command: **Service— Parameters... — System— Files— Auto Save**.

Tab. 9.1.26. Control elements of the auto save setup dialog

Element	Setting description
Smart saving Include	The option serves to enable or disable files auto saving. If auto saving is enabled, autosave files are created at a preset period in the specified folder (see Section 11.3.3.1 on p. 2030). These files can be used to restore documents in case of emergency shutdown (see Section 11.3.3.2 on p. 2030).
Interval of Auto Save, min	The field allows you to type or set with the increment button a time interval value after which the system will do file auto save.
Save to:	Group of elements allows to specify a folder for to automatically save files to. By default, auto save is performed into the system temporary file folder. To specify another folder for auto saving, enable the To Another Folder option, press the Browse... button and select desired folder.

Upon finishing with Auto Save settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.4.5. Save Configuration

You can set up KOMPAS-3D system configuration saving parameters in the **Saving system configuration** dialog that is called by the following command: **Service— Parameters... — System— General— Configuration saving**.

Tab. 9.1.27. Control elements of the save configuration saving options setup dialog

Element	Setting description
Active Window Settings	The group of options makes it possible to select a variant of saving the settings that define the appearance of the main system window. If the saving option is on, then the information about the working window status is stored in the file <i>*.dsk</i> . The list of attributes comprising the information about working window status is shown in Table 9.5.1 on p. 1882. If saving is disabled, the <i>*.dsk</i> file is not created.

Tab. 9.1.27. Control elements of the save configuration saving options setup dialog

Element	Setting description
System Settings	<p>The group of options allows to select a method of system settings saving. System settings include the settings made on the System page of the general dialog for parameter setting up.</p> <p>If the saving option is on, then the information about system settings is stored in the file <i>*.cfg</i>.</p> <p>The file <i>*.cfg</i> is saved on disk, whether or not the system settings are set to be saved or not. In the latter case it only contains the information that saving the settings is disabled. In this case, the existing system settings will be used in the current working session, and the defaults will be used in the next working session.</p>
Settings of New Documents	<p>The group of options allows to select a method of new document settings saving. New document settings include the settings made on the New Documents page of the general dialog for parameter setting up.</p> <p>If the saving option is on, then the information about new document settings is stored in the file <i>*.prj</i>.</p> <p>If saving is disabled, the <i>*.prj</i> file is not created.</p>
Status of Open Documents	<p>The group of options allows to select a method of writing the state of documents opened for editing. The s are available when in the mode of saving the save working window settings mode.</p> <p>If saving the configuration status of open document windows is enabled, then the information is stored in the file <i>*.dsk</i>. During the following KOMPAS-3D session, the same documents will be opened automatically that were opened at the moment of exit from the previous session.</p> <p>If saving the configuration status of open document windows is disabled, then in new KOMPAS-3D working session automatic opening of documents is not performed.</p>

* Interface settings are not recorded in this file (they are recorded in the **.dsk* file), as well as the information that is displayed in the system file and folder location viewing dialog.

Upon finishing with setup of the configuration saving settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

The existing configuration settings can be saved in the profile file (see Section 9.4.1 on p. 1871). Later on, using this profile will make it possible to quickly change the existing settings to the one saved in the profile.



Note that only those settings will be saved in *.cfg*, *.prj* and profile files that are different from the default ones. Information about unchanged parameters is not saved.

When reading *.cfg* and *.prj* files, or applying a profile to the parameters, about which there is no information, default values will be applied. The default path to *SYS* folder will be taken from *KOMPAS.ini* file (see Section 9.5.3 on p. 1883); if such a file does not exist, the default path is determined using the value of the respective variable of KOMPAS environment (see Section 9.5.4 on p. 1886).

9.1.5. Print


9.1.5.1. General settings

General setup of document printing is performed in the **General settings** dialog which is called by the **Service — Parameters... — System — Print — General Settings**.

Tab. 9.1.28. Control elements of the common print settings dialog

Element	Setting description
Number of Print Tasks	<p>Use this field to enter or use a counter to set the number of objects in the list of the recently downloaded or saved print tasks (see Section 8.3.1 on p. 1655). This list is displayed:</p> <ul style="list-style-type: none"> ▼ in menu File— Print Task, ▼ in the menu of Load printing task... button located on the Standard panel, ▼ in the preview mode in the File menu. <p>By default, number of objects in the list is equal to nine.</p>
Size of the list of the latest plotter/printer configuration files	<p>Use this field to enter or set by meter the number of objects in the list of the latest downloaded or saved printer configurations (see Section 8.3.2 on p. 1659). This list is displayed:</p> <ul style="list-style-type: none"> ▼ In the Setup... button's menu of the Print document dialog (see Section 8.1.2 on p. 1631) and of the Special document print dialog (see Section 8.1.3 on p. 1635), ▼ in the File— Print Preview Settings menu, ▼ in the menu of Print preview button located on the Standard panel, ▼ in the File— Plotter/Printer Configuration menu in the preview mode. <p>By default, number of objects in the list is equal to nine.</p>

Tab. 9.1.28. Control elements of the common print settings dialog

Element	Setting description
Default plotter/ printer configurations	<p>The group of elements allows to choose configurations of printing devices which will be used for printing of all special print types of documents by default.</p> <p>For the desired configuration files selection the Browse... buttons are used. Full file names are displayed in the dialog fields.</p> <p>Selected configuration files will be used by default in the next KOMPAS-3D working session.</p> <p>If configuration files are not selected, the default printing device is Windows default printer.</p>
 Browse...	<p>The button allows to select the required plotter/printer configuration file in the standard file selection dialog. A task file has <i>pd</i>c extension.</p>
Full File Path Name in Print Queue	<p>Status of this option affects displaying of the document's file name in the Windows print queue.</p> <p>If the option is enabled, file name in the print queue is formed on the basis of the KOMPAS - <full file name> template, for example: <i>KOMPAS D:\My Documents\Project\Drawing.cdw</i>; if the option is disabled, the following template is used: KOMPAS - <short file name>, for example: <i>KOMPAS - Drawing.cdw</i>.</p> <p>If several documents are sent for print at once (it is possible when working in the preview mode), the document name in the print queue (irrespective of this option) will be formed using the following template: KOMPAS - Print task.hms, where hms is the time the task was sent to print (hours in 24-hour format, minutes, seconds). Example: <i>KOMPAS - Print task 152436</i>.</p> <p>Note that some printers have a limit for the file name length in the print queue. Therefore, this option should only be enabled if it is really necessary to display the file's full name.</p>

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.5.2. Print Filters

The print filter setting up is done in the **Set Print Filters** dialog. This dialog appears on the screen in normal mode of KOMPAS-3D operation after calling the **Service — Parameters... — System — Print — Send to print filters**.

The dialog control elements serve to define document objects to be printed.

Tab. 9.1.29. Control elements of the Print Filter Settings Dialog

Element	Setting description
All Objects	The options serves to enable or disable printing of all document objects. If some and not all of the objects are selected for printing, then the “check” marking the enabled option displays on the gray background. Clicking the option in such a status will disable it. Next — will enable it.
Text As Placeholder	Option enabling means that outline boxes will be printed instead of text
Raster Objects as Placeholders	Option enabling means that outline boxes will be printed instead of raster objects.
Display OLE-objects as placeholders	Option enabling means that outline boxes will be printed instead of OLE-objects.
Apply Filters to Macro Elements and to View and Fragment Inserts	Use this option to control the application of filters to objects included in the macro elements and in the embedded views and fragments. If the option is enabled, filters will be applied to these objects. When the option is disabled, these objects will not be considered independent and subject to filtering.
List of Current Graphic Document Elements	The list contains the names of document elements and their groups. To expand a group list, click the “+” sign before its name. You can change an object print set by switching on and off corresponding options in the list.
Don't Use Filters	The option allows to cancel filters application and print all objects.

To employ the created filters and close the dialog, click the **OK** button. To close the dialog without saving changes, press the **Cancel** button.



The same as the above dialog can be called when working in the preview mode, as well as from the print setup dialog. In these cases, the dialog contains the **Apply** button that makes it possible to make use of created filters without closing the dialog. Filters that are created in the preview mode or in the print setup dialog do not affect system settings of output filters.

9.1.6. Common for documents

9.1.6.1. Thread parameters

You can manage the list of standard thread tables and their connection using **Thread parameters**. This dialog is called by the command **Service — Parameters... — System — Common for documents — Thread parameters**.

The **Thread standards** table includes a default list of files with parameters of a standard thread. The **File name** column shows the file path and the **Thread type column shows the** thread type and name of the standard.

You can add standard threads' parameters to the list of files. Click **Add** to enter a new file to the table. After clicking the button, the standard dialog **Select files to open** appears on the screen.

Select the required file (should have the *xml* extension) and click **Open**. A line for the selected file will appear in the **Thread standards** table.

Enable the option to the left of the file name to attach a file. During hole construction, only those views of standard threads are available that are connected to the system.

If you want to exclude any thread view in the table from construction, disable the option to the left of the file name.

Use the arrow button **Move up/down** to move the line with the file name and thread type in the table.

To delete a file with thread parameters from the list, select the line with its name and click **Delete**. Upon finishing the settings, click the **OK** button. Press the **Cancel** button to close the dialog without setup changes.

9.1.7. Graphic Editor

9.1.7.1. Cursor

Setup of the cursor view is performed using the **Cursor setup** dialog which is called by the **Service — Parameters... — System — Graphic Editor — Cursor** command.

Tab. 9.1.30. Control elements of cursor settings dialog

Element	Setting description
Standard	Group of elements serves to set up the standard cursor parameters (the one displayed as a square catcher).
"Catcher" Size	An element serves to modify the cursor size using a slider. In this case, the current cursor size is displayed in a special viewing window. The cursor size affects its "sensitivity zone": the cursor can select or pick the elements whose portions are within the inscribed circle of the square cursor "catcher", while a dynamic snap activates if the snapping point lies within the circle.

Tab. 9.1.30. Control elements of cursor settings dialog

Element	Setting description
Text frame	Activate this option to have the white 1 pixel wide field displayed around text symbols (e. g. snap name) located near the cursor*.
Borders	Activate this option to have the white 1 pixel wide field displayed around cursor lines*. This makes the cursor more contrasting when located above objects of similar color.
Cursor Color	Group of elements allow to select a standard cursor display color: Inverse or Color . When you choose the Color option, the Color button becomes available that allows to select the desired cursor color. After clicking the button, the Standard Color Selection dialog appears on the screen.
Large	Group of elements allows to set up parameters of large cursor.
Include	To switch to a large cursor size (a large crosshair like coordinate axes), choose this option. For quick switching of cursor appearance (without addressing to the setting up dialog) the combination of <Ctrl>+<K> buttons is used.
Color	Button serves to select the desired color of the large cursor. After its clicking, the Standard Color Selection dialog appears on the screen.
Step	Group of elements allows to modify the default values of the cursor steps, delete any of them, or add new values to the List . During the input, the added values are automatically checked for correctness. To specify the default step value, open up the list By default and select the desired item.
Display Commands Parameters	The option controls the display of command options next to the cursor. If it is enabled, then when creating and editing a geometrical object, the parameter values of this entity are displayed next to the cursor. For example, in case of a line segment, the length and the inclination angle will be displayed.
Rounding	Option serves to enable and disable rounding mode (see Section1.4.2.14 on p. 96) for new document windows. This mode of the current window is controlled by the Rounding button on the Current Status bar.

* The text frame shall be visible in the preview window when the Color of Working Field Background is not white. Working field background color setting up is described in Section9.1.3.1 on p. 1678.

Upon finishing with the cursor settings, click the **OK** button. To leave the dialog without making changes to the cursor settings, click the **Cancel** button.

9.1.7.2. Grid

Setup of the auxiliary grid displayed on the screen is performed in the **Grid setup** dialog which is called by the **Service — Parameters... — System — Graphic Editor — Grid** command.

The same dialog is used for setting the model editor grid and the active window.

Tab. 9.1.31. Control elements of the grid setup dialog

Element	Setting description
Parameters Tab	
Type	Group of options allows to set up grid type. For desired type selection, enable its respective option.
Step along X-Axis, mm	The field serves to customize the distance (in millimeters) between the grid nodes in the X axis direction of the current coordinate system.
Step along Y-Axis, mm	The field serves to customize the distance (in millimeters) between the grid nodes in the Y axis direction of the current coordinate system.
Rotation Angle, deg.	The field allows to set the rotation angle of the grid with respect relative to the X axis of the current coordinate system (in degrees). The angle is calculated from the positive direction of the X axis counterclockwise.
Distortion Angle, deg.	The field allows to set the distortion angle in degrees between the sides of a grid cell which defines the cell distortion (non-orthogonality). By default it is set equal to 90 degrees (rectangular cell).
Isometry	Clicking this button automatically sets the rotation angle equal to 150°, and the distortion angle equal to 60 degrees. A grid with such parameters is convenient to use for drawing isometric images.
Rectangular	Clicking this button automatically sets the rotation angle and the distortion angle equal to 90 degrees (a rectangular grid not rotated with relative to the X-axis of the current coordinate system).
Display Tab	
Point Size, pxl	The field serves to set the grid point size in screen units (pixels). Only odd numbers are allowed.
Color	The button allows to choose grid point color in the standard color selection dialog.
Node Size, pix.*	The field serves to set the node size in screen units (pixels). Only odd numbers are allowed.
Color*	The button allows to choose grid node color in the standard color selection dialog.
Display Node as "Criss-Cross"*	Activate this option to have the grid nodes displayed as "crosses". When the option is disabled, the grid nodes are shown as little squares.

Tab. 9.1.31. Control elements of the grid setup dialog

Element	Setting description
X-Axis Spacing of Nodes*	The field serves to define how many grid points to skip per node in the X direction of the current coordinate system.
Y-Axis Spacing of Nodes*	The field serves to define how many grid points to skip per node in the Y direction of the current coordinate system.
Step of Resampling Down	Group of options sets the resampling (drop) rate of grid points when those cannot all be normally drawn (in a window with a small display scale).
From Series	Activate this option to have a drop rate of displaying grid points selectable from the following set of numbers: 2, 5, 10, 20, 50, 100, 200, 500, 1000, 2000, 5000, 10000, 20000, 50000, 100000, 500000. This means that when you set the display scale not allowing to that does not permit correctly displaying every point, there will be displayed every second grid point will be displayed; with further scaling down— every fifth, then tenth, etc.
Ratio	The option sets the step of dropping the grid nodes as a multiple of any integer number. For example, setting a multiple of 2 as the resampling down step means that upon decreasing the image scale, every second point of the grid will be shown first, then every fourth one, then every sixth, and so on.
Minimum Point Spacing, pix.	The field serves to set minimal distance (in pixels) between grid points. When decreasing the image scale, the distance (in pixels) between the grid points on the screen is decreased. As long as this distance is greater than the minimal specified distance, no grid nodes are dropped. Upon the continued decreasing of the scale, the grid is resampled down according to the specified parameters.

* The element is available if one of the grid types with nodes has been selected on the **Properties** tab.

Upon finishing with the grid settings, click the **OK** button. The grid image in the active window will be redrawn according to the specified parameters. To leave the dialog without making changes to the cursor settings, click the **Cancel** button.

9.1.7.3. Scroll Bars

Controlling the scroll bar display and the image auto shift in the window is done in the **Scroll Bar Setup** dialog called with the **Service— Parameters... — System— Graphic editor— Scroll Bars** command.

The similar dialog is used in scroll bars setup.

- ▼ text editor,

- ▼ BOM editor,
- ▼ model editor,
- ▼ reports,
- ▼ current window.

Tab. 9.1.32. The scroll bar setup dialog control elements

Element	Setting description
Horizontal Ruler	The option allows enabling the display of the horizontal scrollbar.
Vertical Ruler	The option allows enabling the display of the vertical scrollbar.
Auto Scroll	Group of elements allows to set up parameters of auto scroll. Auto Scroll occurs if during some command execution the cursor moves beyond the working field limits while the left mouse button is pressed. The image in the window moves in the direction of the cursor movement by the distance specified in the auto-scroll settings.
Enabled	Option allows auto scroll enabling in the current window.
Of Window Size, %	The field allows to enter manually or set with a counter the distance to shift the image in the window. The distance is set as a percentage of the window size.

Upon finishing with scrollbar and auto-scroll settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.4. System Lines

Use the **System line styles** dialog to set the parameters of display and printing line styles. It appears on the screen after calling the **Services— Options... — System — Graphic editor — System lines** command.

The dialog serves to define the color, the width on screen, and the width in print of the system line styles (those having standard KOMPAS-3D support).

Tab. 9.1.33. Control elements of the system lines setup dialog.

Element	Setting description
Thickness	The group of interface management items serves to set the width of drawing normal, thin and thick lines for drawing. To define the line weight used for display, enter or select the desired number of pixels in the On Screen field. To define the line weight used for printing out, enter the desired value (in millimeters) in the On Paper field.

Tab. 9.1.33. Control elements of the system lines setup dialog.

Element	Setting description
List of Lines	<p>This list contains all KOMPAS-3D system lines. It is laid out as a table of three columns: Color, Image, Name.</p> <p>To change a line color, click the mouse in the desired table row. Then click the black triangle button that appears in the Color cell and specify the required line color. The selected color will be used for both displaying and printing the lines of the respective styles.</p> <p>With changing the color, as well as the weight (thickness), of the displayed line, its appearance in the Image entry also changes.</p> <p>Line names cannot be edited.</p>

Upon finishing line style settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.5. System Symbols

Color of system characters (which are by default supported by KOMPAS-3D) is set in the **System characters color** dialog. It appears on the screen after calling the **Services—Options... — System— Graphic editor— System symbols** command.

To select a symbol color, click the button with its name. In the appearing standard file selection dialog, specify the desired color.

The symbols appear in a preview windows to let you examine evaluate the performed changes.

Upon finishing symbol color settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.6. Ghosts

Ghost is an image that temporarily appears on the screen when performing any operation, that shows the current state of objects being created or edited. For example, when creating a straight line segment, then the line ghost is displayed while awaiting for the second point to be fixed. As the cursor moves, the ghost is rebuilding dynamically showing the current state of the line being created.

Different object construction variants are shown as ghosts for the same input data. For example, two tangent circles can be laid through one point. In this event, one ghost is active and the newly created object will correspond to it; other ghosts are passive. To create an object, the user can activate any ghost. By default, the active ghost is displayed in a solid black line, while passive ones — in a dotted line.

Setting the appearance of phantoms that are displayed on the screen is the setting of drawing of lines that show the form and position of newly created or edited objects prior to their being fixed. This customization is made in the **Ghost drawing** dialog activated by the **Service — Parameters... — System — Graphic Editor — Ghosts** command.

Tab. 9.1.34. Control elements of the ghost displaying setup dialog

Element	Setting description
Active	The option serves to activate the setup of the active ghost parameters.
Non-active	The option serves to activate the setup of the passive ghost parameters.
Style of active/ passive ghost	<p>Group of elements serves to choose line style for ghost (active or passive) displaying. The following methods are available:</p> <ul style="list-style-type: none"> ▼ Solid, ▼ Dotted, ▼ Dashed, ▼ By Object— a ghost line style match the display style of the emerging object. <p>When selecting By Object option, the weight and color of lines setup is impossible.</p>
Thickness	The field serves to set the line weight (width, in pixels) to display the ghost on the screen. To have the ghost line weight match the weight of the lines of the emerging object being modeled, activate the By Object option.
Color	The button allows to select a ghost line color. It calls the standard Windows color selection dialog. To have the ghost line color match the color of the emerging object being modeled, activate the By Object option.

The external appearance of the active and passive ghost lines appears in the respective preview windows to let you examine the performed changes.

Upon finishing with ghost settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.7. Constraints and degrees of freedom

Use the **Limitations and range of discretion** dialog to set the parameters of displaying characters that indicate limitations and range of discretion. It appears on the screen after calling the **Services— Options... — System — Graphic Editor — Constraints and Degrees of Freedom** command.

The setting made in this dialog affects sketches and graphic documents.

Tab. 9.1.35. Control elements of the degrees of freedom and constraints setup dialog

Element	Setting description
Restrictions	The group of elements serves to set up the display of relations and constraints applied to graphical objects. The list of constraint symbols is presented in Section 7.2.5.1 on p. 1620.
Color	The button serves to change the color of constraint symbols displaying. After clicking the button, the Standard Color Selection dialog appears on the screen.
Always Display Dimensions with Variables, Fixed Dimensions and Points	<p>The option allows to control the display of the following objects:</p> <ul style="list-style-type: none"> ▼ Variables assigned to dimensions ▼ Frames surrounding fixed dimensions ▼ Fixed points symbols. <p>If the option is enabled, these objects are displayed on the screen regardless of whether the display of constraints is enabled or disabled. If the option is disabled, these objects are shown only when the constraints display is on.</p>
Degrees of Freedom	The group of elements makes it possible to adjust the color of symbols indicating the range of discretion of graphical objects. The list of degrees of freedom symbols is presented in Section 7.2.5.2 on p. 1624. The group contain only Color button that call the standard color selection dialog.

The external appearance of the constraints and degrees of freedom icons appears in the respective preview windows to let you examine the performed changes.

When finished with setting up symbols display, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.7.8. Views



The display of background and hidden views can be set up in the **View Display** dialog. It appears on the screen after calling the **Services— Options... — System — Graphic editor — Views** or clicking the **View setup** button in the **Document manager** (the **Document manager** dialog is described in section 3.5.6.2 on p. 1182).

Tab. 9.1.36. Control elements of the background and hidden views display setup dialog

Element	Setting description
Show View Names	The option manages view name display. If the option is enabled, view names and numbers are displayed in the Current View field of the Current Status toolbar (see Fig.3.5.13 on p. 1171), if the option is disabled— only view numbers.

Tab. 9.1.36. Control elements of the background and hidden views display setup dialog

Element	Setting description
Background Views* Frames of Hidden Views Borders of Associative Views	The options make it possible to select a variant of the objects that can be adjusted.
Show	The option manages the display of the borders for hidden and associative views. It is available in the dialog when the Frames of Hidden Views or Borders of Associative Views is enabled.
Thickness	The field allows to set the desired width (in pixels) to display the object on the screen. The value is entered into the field manually or specified using the counter. The field is available for the Solid line style.
Line	Group of options allows to select the line style for object display. To select style, enable its respective option— Solid , Dashed or Dotted .
Color...	The button allows to specify the display color of the object. It calls the standard Windows color selection dialog.

* This setting affects the object line style in the macro element editing mode.

The objects being set up appear in the respective preview windows. This helps evaluate the introduced changes.

Upon finishing with view display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.9. Layers

The display of background and hidden drawing layers can be set up in the **Layout Display** dialog. It appears on the screen after calling the **Services— Options... — System — Graphic editor — Layers** or clicking the **Layer setup** button in the **Document manager** (the **Document manager** dialog is described in section 3.5.6.2 on p. 1182).

Tab. 9.1.37. Control elements of the layer display setup dialog.

Element	Setting description
Show Layer Names	The option manages layer name display. If the option is enabled, view names and numbers are displayed in the Current Layer field of the Current Status toolbar (see Fig.3.5.21 on p. 1189), if the option is disabled— only numbers.

Tab. 9.1.37. Control elements of the layer display setup dialog.

Element	Setting description
Background Layers	The window serves to preview the appearance of the objects to be set up. This helps evaluate the introduced changes.
Thickness	The field allows to set the desired width (in pixels) to display the object on the screen. The value is entered into the field manually or specified using the counter. The field is available for the Solid line style.
Line	Group of options allows to select the line style for background layer display. To select style, enable its respective option— Solid , Dashed or Dotted .
Color...	The button allows to select background layer display color. It calls the standard Windows color selection dialog.

Upon finishing with the layer display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.10. Coordinate systems

The style of coordinate system axes display parameters setup is performed in the **Coordinate System Axes Display** dialog. It appears on the screen after calling the **Services—Options... — System— Graphic editor— Coordinate systems** command.

Tab. 9.1.38. Control elements of the coordinate system setting dialog

Element	Setting description
Axes of Local Coordinate System	The window serves to preview the appearance of the objects to be set up. This helps evaluate the introduced changes.
Show	Option is controlling the way the coordinate system axes are displayed on the screen *.
Thickness	The field allows to set the thickness (in pixels) of the coordinate system axes displayed on the screen. The value is entered into the field manually or specified using the counter. The field is available for the Solid line style.
Line	Group of options allows to select the line style for coordinate system axes display. To select style, enable its respective option— Solid , Dashed or Dotted .
Color...	The button lets you configure the display color of the coordinate system axes. It calls the standard Windows color selection dialog.

* The coordinate system axes are not printed.

Upon finishing with coordinate axes display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.11. Editing

The graphic document object parameters setup is done in the **Edit** dialog called by the **Service— Parameters... — System— Graphic Editor— Edit** command.

Tab. 9.1.39. Control elements of the edit options setup dialog

Element	Setting description
Selection	<p>The button allows to customize color of the selected objects. You often need to select objects before running a command. For example, to obtain a copy of a graphic object, the source graphic object should be selected; to create a forming 3D element, its sketch should be selected.</p> <p>After clicking the button, the Standard Color Selection dialog appears on the screen. A corresponding preview window allows you to assess made settings by sight.</p>
Highlighting	<p>The button allows to customize color of the specified objects. Objects must be selected in order to perform certain commands. For example, to construct a swept feature specify its path; to construct a parallel line— other line segment or straight line.</p> <p>After clicking the button, the Standard Color Selection dialog appears on the screen. A corresponding preview window allows you to assess made settings by sight.</p>
Show Hidden Layers of Selected Complex Objects	<p>Enable this option for visibility of elements in layers the drawing of which is disabled at the moment when selecting complex objects (for more information about layer state management, see Section 3.5.6.5 on p. 1189).</p> <p>Complex items include groups (see Section 3.9.2 on p. 1280), fragment insertions (see Section 3.7.2 on p. 1248), macro elements (see Section 3.7.5 on p. 1254).</p>
Always show a fill in the specified color	<p>Enable this option to highlight only the fill contour and not to change its color, when selecting the fill.</p> <p>If the option is disabled, then when you highlight the fill, its color changes to the color that set for selected objects.</p>

Tab. 9.1.39. Control elements of the edit options setup dialog

Element	Setting description
Always show figures in source colors	<p>Enable this option to highlight only the fill contour and not to change its color, when selecting the fill.</p> <p>If the option is disabled, when the drawing is highlighted, its colors will be replaced by the color shades preset for the highlighted objects.</p> <p>Note that changing the method of displaying the highlighted images is possible, if OpenGL[*] is used for drawing the graphical documents. If for document drawing GDI^{**} is used, the pictures are always shown in the source colors; in this case, the option is unavailable.</p> <p>Selection of a image generation method— OpenGL or GDI— is carried out in the view control dialog (see Section 9.1.7.17 on p. 1719).</p>
Zooming Factor	<p>The field allows to enter the zoom in or the zoom out factor for the image in the window that will be imposed upon pressing the <SHIFT>+<+> и <SHIFT>+<-> keys, respectively.</p>
Mouse Movement Resolution, mm	<p>The field serves to define the minimum amount of the mouse movement (for example, when moving selected objects), after which the system starts responding to the movement. This setting is introduced to prevent random mouse movements (e.g. when your hand trembles, your table is pressed, etc.) and accidental actions with drawing objects. The mouse movement resolution (dead zone) is specified in millimeters, and the current image zoom scale does not affect it.</p>
Undo Steps	<p>The field serves to enter the number of operations that can be undone or redone when working. The value is entered into the field manually or specified using the counter.</p> <p>When increasing this value, the allocation of system resources (memory) to maintain the list of performed operations is increased, therefore, the number of steps must be justified. It is recommended to set up this number in the range of 10 to 30.</p>
Group Similar Actions	<p>Enable this option to have several consecutive object scaling or selection actions to be stored in the Undo list as a single action. In this case, all grouped operations will be undone or redone as one. For example, you have successively scaled the same objects. If the grouping option was enabled, then clicking the Undo button should restore the state existing prior to the very first scaling of objects.</p>

* OpenGL (Open Graphics Library) is the specification of the program interface for applications that uses 2D and 3D computer graphics.

** GDI (Graphics Device Interface or Graphical Device Interface) — is the Microsoft Windows interface for graphic objects presentation and their render to display units, such as monitors and printers. Draws lines and curves, displays fonts and processes palettes.

Upon finishing with editing settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.12. Handles

Display of handles is set up in the **Handles** dialog. It appears on the screen after calling the **Services—Options... — System — Graphic editor — Handles** command.

The **Handles** dialog contains:

▼ The **Number of selected objects for displaying handles, max** field

This field is used for setting the maximum number of selected objects. When these objects are selected simultaneously, their handles are displayed. The number of selected objects defined in percentage from 0 to 100.

If the number of selected objects exceeds the specified value, their handles are not displayed. Therefore, when 0 is entered, handles will never show.

▼ The **Display only general defining points for group of selected objects** option

If the option is enabled, only those handles are displayed that are common for selected objects.

If the option is disabled, all handles of selected objects are displayed.

Once you have finished setup of handles display parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.13. Bitmap Objects Imported into Document

The bitmap object editing options imported into graphic documents are customized in the dialog **Editing of Bitmap Objects Introduced into Document**. It appears on the screen after calling the **Services—Options... — System — Graphic Editor — Bitmap Objects Imported into the Document** command.

Tab. 9.1.40. Control elements of the bitmap objects embedded in a document setup dialog

Element	Setting description
Type of File for Editing	The drop-down list allows to select the format to use for the graphic information when editing.
According to Windows-registered File Type	Selecting this option means a program will be launched to edit pictures, which is assigned by Windows to the file type specified in the Type of File for Editing list.
Other	Enable this option to choose other program to edit pictures. The full path to the program file is displayed in the info field. To change the program, click the “...” button at the right of the input field. The mentioned program must support the file type selected in the Type of File Editing list.

Tab. 9.1.40. Control elements of the bitmap objects embedded in a document setup dialog

Element	Setting description
Save Original Bitmap Resolution	<p>Activating this option means changes to the picture resolution due to editing will be ignored. Due to this, the picture size after editing will be the same as before.</p> <p>Some programs, for example, MSPaint, automatically assign certain resolution when saving the images that do not carry the resolution data within themselves (when inserting such pictures in a KOMPAS document, you can set an arbitrary resolution).</p>

To finish the setup, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.7.14. Simplified Display

Simplified display is used in the cases when the current image scale becomes too small to show document details. Parameters for simplified displaying of objects on the screen are set up in the **Simplified drawing** dialog called by the **Service— Parameters... — System— Graphic editor— Simplified drawing** command.

Tab. 9.1.41. Control elements of the simplified display setup dialog.

Element	Setting description
Minimum Height of Text Line on Screen, mm	<p>The field allows to define the minimum height of text characters (in millimeters) to be displayed on the screen. If the text height on the screen becomes less than the specified value, the text will be displayed as an outline box.</p> <p>The value is entered into the field manually or selected from the predefined list.</p>
Image Zoom Image Size below which All Lines Are Drawn Solid and without Arrows	<p>The field allows to define the image size display scale below which the simplified line display takes place (all lines will be drawn as solid and without any arrows).</p> <p>The value is entered into the field manually or selected from the predefined list.</p>
Draw Outline Rectangles for Raster Objects	<p>Activating this option means the outline rectangles will be used in place of the bitmap objects inserted in graphic documents.</p>
Draw Outline Rectangles Instead of OLE-Objects	<p>Activating this option means the outline rectangles will be used in place of the OLE-objects inserted in graphic documents.</p> <p>Enabling outline box display instead of bitmap and OLE-objects helps save document loading time.</p>

Tab. 9.1.41. Control elements of the simplified display setup dialog.

Element	Setting description
Always draw a fill in the specified color	<p>The enabled option means all fills in documents appear in colors set for them regardless of the state of layers and views they are located on. If the option is disabled, only fills that are on the current layer appear in their own color, while other fills acquire the color of layers or views that contain them.</p> <p>Please note that this setting applies to printing documents and saving them in bitmap format. For example, if a color set for the view was selected for the print of a drawing, then if the Always draw the fill with set color option is enabled fills will be printed in their own colors, if disabled — in colors of views they are located in.</p>

Upon finishing simplified display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.15. Targeting

Several KOMPAS-3D commands use dynamic object targeting. This means, that as the cursor moves over an object which can be selected in the given command, the object is highlighted. If several objects that can be used in the command are located close to each other, dynamic targeting helps select the desired object. By clicking the mouse, you can select the object that is currently highlighted.

Dynamic targeting of objects, when enabled, allows indicating (selecting) any of the nearby located objects (including overlapping ones). To pick over closely located objects, use the appropriate commands in the context menu. These commands are available if several objects are in the cursor's catcher when specifying (or selecting) an object.

Dynamic object search enabling and disabling is performed in the **Dynamic Object Search Setup** called by the **Service— Parameters... — System — Graphic Editor — Targeting** command.

The only option in the dialog serves to enable or disable dynamic object targeting.

Once the desired option is set, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.16. Snaps

Global Snaps Management is performed in the **Set up Global Snaps** dialog. This dialog appears on the screen in the following cases:

- ▼ after calling the **Service — Parameters... — System— Graphic Editor— Snaps** command.
- ▼ after calling the **Set up Parameters** command from the **Snaps** button menu located on the **Current Status** or **Global Snaps** bar.



In the first case, the setting you made is saved as default and used for all newly opened and created documents. In the second case, the setting made is used only in the Windows of the current document until the end of its current working session.

Tab. 9.1.42. Control elements of the snap setup dialog

Element	Setting description
List of Global Snaps	The list allows to change a set of global snaps active in the document. To do this, enable and disable options that correspond to the required names in the list.
Move up Move down	Buttons are used to set up the snap priorities. If several global snaps are enabled and all of them can act at the current cursor position, then the only one snap with the higher priority operates. A snap priority is determined by its position in the list. To increase a snap priority, select it in the list and click the Move Up button, and to decrease it, press the Move Down button. The selected snap will move by one position in the specified direction.
All Snaps	Option allows to enable or disable all snaps at once. If not all snaps are enabled, then the “check” marking the enabled option is displayed in the gray background. Clicking the option in such a status will disable it. Next — will enable it.
Settings affecting both global and local snaps.	
Track Dynamically	Option controls the dynamic calculation of the snaps. If it is enabled, the selected snaps are calculated on the fly— when the cursor moves towards a point that can be snapped (it is marked on the screen with a ghost in the form of a cross. *). When this option is disabled, calculation is performed only after you fix a point.
Show Text	Option allows to display the activated snap name next to the cursor. The snap name will be displayed if Dynamic tracking is enabled. For an angular snap, the angle value is also shown additionally. If the option is disabled, when you hover the cursor over the point that can be a snap point, no name of the snap or ghost in the cross form will be displayed.
Snap to a background layer objects	The option serves to make a snap to objects lying on the background layers (see Section 3.5.6.1 on p. 1181) of graphic document or sketch.

Tab. 9.1.42. Control elements of the snap setup dialog

Element	Setting description
Only to visible grid points	Use this option to use only knots of the grid when snapping By grid . It is available, if the set of active snaps in the document includes the By grid snap (the respective option is enabled in the list of global snaps). When this option is enabled, grid snapping will be applied to grid knots only and will be possible only if the grid is displayed on the screen (see Section 3.1.4.1 on p. 885). When the option is disabled, the snapping to the grid points that become invisible when dropped (see Section 3.1.4.3 on p. 886) is available. Enabling grid display is not necessarily.
Angular snap step	The field allows to enter the value with respect to which the angular snap angle should be applied. For example, if the increment of the angular snap is set to 15°, snapping will be possible to points located on lines passing through the last fixed point at angles of 15°, 30°, 45°, 60°, 90°, etc.
Snapping to the drawing layout elements	You can use this option to snap to lines of a drawing's layout (frame, table of the title block).
Snapping to model elements	Option allows to make a snap to model elements when working with sketch Snapping to model elements is detailed in Section **.
Only to visible model elements	Option allows to use only visible objects when making a snap to a model objects Available when the Snapping to model elements option is enabled.
Prohibit snaps	Option allow to enable or disable all global snaps in the current window. It is available in the dialog if called by Set up Parameters command from Snaps button menu.

* The color of the ghost and the text corresponds to the color set for the enlarged cursor.

**

Upon finishing global snap settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.7.17. Image Management

Method of generating a drawing or fragment on the screen setup is done in the **Image Management** dialog called by the **Service — Parameters... — System — Graphic Editor — Display Control**.

For document drawing OpenGL or GDI may be used.

OpenGL (Open Graphics Library) is the specification of the program interface for applications that uses 2D and 3D computer graphics.

GDI (Graphics Device Interface or Graphical Device Interface) — is the Microsoft Windows interface for graphic objects presentation and their render to display units, such as monitors and printers. Draws lines and curves, displays fonts and processes palettes.

The **Image control** dialog contains two options— **Use OpenGL** and **Use hardware acceleration**.

- ▼ If the **Use OpenGL** option is enabled, for document drawing OpenGL is used, if disabled— GDI.

The enabled **Use OpenGL** option allows to accelerate the image generation, e.g. at a change of scale or at image shifting. Acceleration is especially noticeable when working with dense drawings.

Fonts, solid fills and bitmap objects displayed using OpenGL can have insignificant visual differences from the same objects displayed using GDI, which is not considered to be a defect. When generating an image in the print preview and at printing, OpenGL is not used, therefore this option does not influence on receiving hard copies of the documents.



Displaying of vector fonts including fonts GOST type A (plotter), GOST type B (plotter), Symbol type A (plotter), Symbol type B (plotter) supplied with KOMPAS-3D using OpenGL is impossible.

Some video cards may fail to support generating the image using the OpenGL interface. For additional information ref. to *ReadMe.pdf* file. To open it, the **Start — Programs — ASCON — KOMPAS-3D — Installation Information** command can be selected in the Windows main menu.

- ▼ The **Use hardware acceleration** option is available, if the **Use OpenGL** option is enabled. Enabling the **Use hardware acceleration** option means that the graphical document image on the monitor screen is formed by the graphics processor. If the document image is poor, disable the **Use hardware acceleration** option. After this, the image will be formed by the central processor (which may slow down the computer's performance).
If your PC has a graphics card that does not support hardware acceleration, this option will be disabled and unavailable.
To apply the image generation changes after enabling or disabling this dialog options, KOMPAS-3D shall be restarted.

9.1.8. Text Editor

9.1.8.1. Scroll Bars

Controlling the scroll bar display and the image auto shift in the window is done in the **Scroll Bar Setup** dialog called with the **Service— Parameters... — System— Text editor— Scroll Bars** command. Working with the dialog is described in Section 9.1.7.3 on p. 1706.

9.1.8.2. Editing

The KOMPAS-3D text editor parameters setup is done in the **Edit** dialog called by the **Service — Parameters... — System— Text Editor— Edit** command.

Tab. 9.1.43. Control elements of the text editor setup options dialog

Element	Setting description
Zooming Factor	Enter the zoom in or the zoom out factor for the image in the window that will be imposed upon pressing the <i>SHIFT</i> >+<+> and < <i>SHIFT</i> >+<->, respectively.
Undo Steps	Enter the number of operations that can be undone or redone when working. When increasing this value, the allocation of system resources (memory) to maintain the list of performed operations is increased. The number of steps must be justified. It is recommended to set up this number in the range of 10 to 30.
Group Similar Actions	To have similar operations saved in the undo list as one, activate this option. In this case, all grouped operations shall be undone or redone as one. Examples of similar-type actions are selecting a series of objects or a series of sequential moves of the same group of elements.
Thickness	Set the desired width (in pixels) to display the outline frame on the screen. The setting is available for the solid line style.
Color	To call the dialog for changing object colors, click the Color button.
Line	Choose the desired line style (solid , dashed or dotted) to display an object on the screen.

The appearance of the adjustable outline frame of the text is displayed in the view window to let you examine the performed changes.

To finish the setup, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.8.3. Text Templates

Setup of text templates parameters is performed using the **Text patterns** dialog which is called by the **Service — Parameters... — System — Text Editor — Text Templates** command.

Tab. 9.1.44. Control elements of the text template setup options dialog

Element	Setting description
Text Template File Name	This field displays a full path to the file which contains text templates. File name by default is <i>graphic.tdp</i> .

Tab. 9.1.44. Control elements of the text template setup options dialog

Element	Setting description
Browse...	The command serves to change a file of text templates. After clicking it, the standard File Selection dialog will appear.
Save Entered Text to File	Option allows to automatically save entered into document text to the text template file (while filling in a title block, a text part of BOM items, typing BOM names on a sheet).
Font	A drop-down list allows to select a font to display template texts in the Text Template Library File window. The list contains all fonts enabled in Windows.
Height, mm	The field serves to enter or select from the preset list the height of text template symbols (in millimeters). Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.
Italic	The option allows to set the italic style of characters.
Bold	The option allows to set the bold style of characters.
Color...	The button serves to set up the symbol color. It calls the standard Windows color selection dialog.
Example	The viewing window is used to display the adjusted text. This helps evaluate the introduced changes.

Upon finishing with display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.8.4. Line Width of Special Symbols

Line width setup to output (whether on the screen or in print) the special symbols (i.e. surface finish symbols, the symbols of weld joints, etc.) is done in the **Line width of special symbols dialog**. It appears on the screen after calling the **Services— Options... — System — Text Editor — Line width of special symbols** command.

Tab. 9.1.45. Control elements of the line width of special symbols setup dialog.

Element	Setting description
System Settings of Line Weight	To have the special symbol line weight match the weight of the normal or thin system line, enable this option. In the field enter the font height (in millimeters), at which the used system line will be switched. By default, this value is 10 mm. If the font height is less than 10 mm, then the special symbols will be drawn in thin lines. If the height is greater than 10 mm, then the special symbols will be drawn in the normal style.

Tab. 9.1.45. Control elements of the line width of special symbols setup dialog.

Element	Setting description
User Setting of Line Weight	To have the special symbol line weight proportional to the font height, enable this option. In the field, enter the ratio (in percent) of the font height and the line width of Special Symbols. By default the line weight is equal to 7.14% (1/14) of the font height.

Upon finishing with special symbols line weight display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.8.5. Zoom for Editing

You can set the default scale, in which text and tables to be edited (including title blocks) will open in the graphic documents, and tables in text documents of the KOMPAS-3D system. This customization is made in the **Text object editing scale** dialog activated by the **Service — Parameters... — System— Text Editor— Editing scale** command.

Tab. 9.1.46. Control elements of the zoom for editing options setup dialog

Element	Setting description
Current Document Zoom	Make this to have the document image scale changing when calling the commands to edit text and tables by enabling this option.
Another Zoom	Make this to have the document image scale changing when calling the commands to edit text and tables by enabling this option. Enter the scale factor value in this field or select the value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.

Example. Current scale of the drawing is 1:5. The **Another scale** option is enabled and the scale factor of *1.0* is entered in the dialog. Upon double-clicking on the drawing's title block, the KOMPAS-3D system goes into the editing mode. The title block will be shown on the screen in the actual size (to the 1:1 scale). After finishing editing the title block, the image scale 1.5 will be restored.

Upon finishing text document editing scale settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.8.6. Spelling Options

Setting up spelling check parameters is done in the **Spelling Options** dialog.

It appears on the screen after calling the **Services— Options... — System — Text editor — Spelling options** or clicking the **Parameters** button in the **Spelling** dialog.

Spelling is checked for all languages, grammar— for the Russian language only.

Tab. 9.1.47. Control elements of the spell checking setup dialog

Element	Setting description
Spelling group	
Check Spelling Automatically	This option allows to enable automatic spell checking (see Section 4.1.3.2 on p. 1322). The words suspected to have errors will be underlined in the text with a waves line. By default, the line color is red.
Select only from the Basic Dictionary	This option serves to disable the use of additional dictionaries (see Section 4.1.3.6 on p. 1330) when displaying the correct spelling variants.
Ignore Words in UPPERCASE	The option serves to skip words in all capital letters during spell check.
Ignore Words with Numbers	The option serves to skip words combined of letters and digits during spell check.
Ignore Fractions, Indices and Superscripts	The option serves to skip fractions, indices, subscripts and superscripts during spell check.
Ignore Internet addresses and File Names	This option serves to skip Internet and e-mail addresses and file names during spell check.
Dictionaries...	This command serves to do various actions with auxiliary dictionaries, for example, create, edit, etc. Once this button is pressed, the screen displays Auxiliary Dictionaries dialog.
Grammar group*	
Check Grammar automatically	This option allows to enable automatic grammar check (see Section 4.1.3.2 on p. 1322). Sentences or part of sentences suspected to have errors will be underlined in the text with a waves line. By default, the line color is green.
Settings...	The button serves to set up the grammar parameters. After calling the command, the Grammar Settings dialog appears.

* The group is not available if spell check tools for the Russian language are not installed (i.e. if the respective option was disabled during KOMPAS-3D installation). In order to install the spell check tools, modify installation parameters.

Upon finishing spelling check setting up, click the **OK** button. To close the dialog and reject changes, click the **Cancel** button.

9.1.9. BOM editor

9.1.9.1. Scroll Bars

Controlling the scroll bar display and the image auto shift in the window is done in the **Scroll Bar Setup** dialog called with the **Service— Parameters... — System— BOM editor— Scroll Bars** command. Working with the dialog is described in Section 9.1.7.3 on p. 1706.

9.1.10. Application Libraries

9.1.10.1. Disabling

You can set up disconnection of application libraries from the KOMPAS-3D system in the **Library disconnection setup** which is called by the **Service — Parameters... — System — Application libraries — Disconnection command**.

Tab. 9.1.48. Control elements of application library detachment setup dialog

Element	Setting description
Auto Detach Libraries Attached in the "Blind" Mode	<p>In some cases, a library is automatically connected to KOMPAS-3D (in the so-called "blind", or hidden mode). The names of thus connected libraries are not displayed in the Library menu. Examples of connecting a library in the "blind" mode are:</p> <ul style="list-style-type: none"> ▼ Running file export/import commands, ▼ Editing parameters of document objects created by an application library, ▼ Calling library commands by clicking the respective button on a User Control panel. <p>To save time on a repeated linking of libraries, when a new call is made to them, by default such libraries are not detached from the system automatically when done working with them is completed. This does not matter when using KOMPAS-3D in a single-user mode. However, when networking with KOMPAS-3D, a "floating" library license on the network HASP key will be locked. In other words, an a library that is unused but not yet detached from the system library will be using up one of the network licenses, preventing that library from running on another computer. Activating this option automatically unloads the libraries "blindly" connected to the system, once their work is done. Another way is to manually detach the library in the Library Manager (see Section 12.1.1 on p. 2035).</p> <p>Make sure that you enable this option if, when networking with the KOMPAS-3D system, the number of licenses for a library (for example, for the library of export into DXF) is less than the number of licenses for the KOMPAS-3D system (for example, 5 licenses for KOMPAS-3D and 2 licenses for the library of export into DXF).</p>
Detach Libraries on Exit from System	<p>Activating this option automatically disconnects all connected libraries of models, fragments and application libraries upon finishing a KOMPAS-3D working session. When the option is disabled, libraries are not detached. Upon the next launch of the KOMPAS-3D system, all connected libraries will be automatically opened in the state in which they were at the time of exiting the system.</p>

Upon finishing with library detachment settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.10.2. Element Editing

The library macroelement handle editing is set up in the **Library macroelement editing** dialog which is called by the **Service — Parameters... — System — Application libraries — Element editing command**.

Tab. 9.1.49. Control elements of the element editing options setup dialog

Element	Setting description
Off	Select this option to disallow editing of the handles of objects regardless of whether the library that sourced those objects is connected or not.
On Only When Library Is Attached	Select this option to edit handles of objects only if the library that sourced those objects is connected to the system.
On	Select this option to edit the handles of objects regardless of whether the library that sourced those objects is connected or not.

Upon finishing with library element editing customization capabilities settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.11. Model Editor

9.1.11.1. Grid

Setup of the auxiliary grid displayed on the screen is performed in the **Grid setup** dialog which is called by the **Service — Parameters... — System — Model Editor — Grid** command. Working with the dialog is described in Section 9.1.7.2 on p. 1705.

9.1.11.2. Scroll Bars

Controlling the scroll bar display and the image auto shift in the window is done in the **Scroll Bar Setup** dialog called with the **Service— Parameters... — System— Model editor— Scroll Bars** command. Working with the dialog is described in Section 9.1.7.3 on p. 1706.

9.1.11.3. System lines

Line width for drawing 3D curves in the model can be set up in the **System styles of lines** dialog which is called by the **Service— Parameters... — System— Model editor— System styles of lines** command.

The dialog contains a table with three columns: **View, Style, Thickness**.

Examples of KOMPAS-3D system line styles are displayed in the **View** column.

The line style names are displayed in the **Style** column.

Thickness value for each style are displayed in the **Thickness** column.

To set the line thickness, activate the required cell in the **Thickness** column by a mouse click and set thickness in pixels.

When changing the thickness, the style example displaying also changes in the **View** column.

The line style name can not be edited.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



Please note that this setting affects only 3D curves (see Section 2.5.4 on p. 360) and does not affect such objects as sketches, edges and axes.



You can set the color of 3D lines. The default color for the lines of different types is set up in the Object Properties Setting Up dialog (see Section 9.2.7.4.7 on p. 1850); if necessary, the color of the created or edited line can be changed on the **Properties** page of the Property Bar.

9.1.11.4. Design Element Libraries

The dialog of design element libraries settings serves to specify the fragment library files **.lfr*, which are used as a sketch libraries and libraries of sketch profiles for round holes when calling the respective KOMPAS-3D commands.

This dialog appears on the screen after calling the **Service — Options... — System — Model Editor — Design Element Libraries** command.

Tab. 9.1.50. Control elements of the design element libraries options setup dialog

Element	Setting description
Name of Hole Library File	The field contains the full name of the hole library file.
Edit...	The button allows to use other file as a hole library. After clicking the button, the standard file selection dialog appears on the screen. Select the required library file. It will be automatically used by the system when calling the Hole from Library command.
Name of Sketch Library File	The field contains the full name of the sketch library file.
Edit...	The button allows to use other file as a sketch library. After clicking the button, the standard file selection dialog appears on the screen. Select the required library file. It will be automatically used by the system when calling the Sketch from Library command.

Specify the library file names and click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.11.5. Image Management

Parameters of displaying models in the window are set up in the **Image control** dialog. It appears on the screen after calling the **Services — Options... — System — Model Editor — View Control** command.

Tab. 9.1.51. Control elements of the image management options setup dialog

Element	Setting description
Step of Model Panning (in % of Window)	<p>Field for entering model movement value in the window with the image-moving key pressed once. The moving increment is set in percent of the window size.</p> <p>For example, after entering the value <i>25</i> in this field, pressing the <Shift>+<▶> keys will move the image to the right by a quarter (<i>25%</i>) of the window width.</p>
Angular Step of Model Rotation (degr.)	<p>Field for entering model rotation value in the window with the image-rotation key combination pressed once. The rotational step is set in degrees.</p> <p>For example, after entering the value of <i>15 degrees</i> in this field, pressing the <Alt>+<▶> keys will cause the part to rotate around the vertical axis by <i>15 degrees</i> to the right (and its image will change accordingly).</p>
Zooming Factor	<p>Field for entering zoom in/zoom out value in the window with the image-zooming key combination pressed once.</p> <p>For example, after entering the value <i>1.5</i> in this field, pressing the <Shift>+<-> keys will make reduce all linear dimensions of the image reduced in $1 \frac{1}{2}$ times.</p>
Use hardware acceleration	<p>This option, if enabled, makes the model image on the screen to be formed by the graphical processor. If the model image is poor, disable the Use hardware acceleration option. After this, the image will be formed by the central processor (which may slow down the computer's performance).</p> <p>If your PC has a graphics card that does not support hardware acceleration, this option will be disabled and unavailable.</p>

Tab. 9.1.51. Control elements of the image management options setup dialog

Element	Setting description
Use additional frame buffer	<p>This option serves to enable and disable the use of additional video card computer hardware capabilities that serve to speed up the three-dimensional model display. Available if the User hardware acceleration option is enabled.</p> <p>By using the frame buffer, you can speed up the redrawing of the previously displayed image[*].</p> <p>For instance, it is possible to speed up restoration of the image part which was hidden under a menu or dialogue box; it is also possible to speed up dynamic targeting (provided that inversion during highlighting is disabled)^{**} due to quicker image restoration during cursor movement from one item of the model to another.</p> <p>The more complex is the model, the more noticeable is the speeding up. Note, however, that not all video cards support these performance capabilities, and not all video card drivers utilize these capabilities. That is why the acceleration performance depends on the video card model and driver version if the Use additional framebuffer option is enabled.</p>
Transparency	<p>Option group option allows to select display method of transparent items (faces, components, surfaces etc.).</p> <p>Enable the Lattice option, so that the area occupied by a transparent item will be filled with separate pixels. The color of pixels corresponds to object color when transparency is equal to zero. Pixels are arranged on screen and they create a regular structure — a grid. “Lattice cells”, i.e., areas where transparent object pixels are not present, are the larger, the higher the Transparency parameter value is for the given item^{***}.</p> <p>Enable the Realistic option to display transparent objects most similarly to objects made of transparent material.</p>

* In some cases, the enabling this option may cause slowing down. In this case, disable it.

** Dynamic search is described in Section 2.1.4.1.2 on p. 131; inversion enabling and disabling— in Section 9.1.11.8 on p. 1732.

*** Transparency is usually set together with defining object properties (see Section 2.14.2 on p. 784). Default value of **Transparency** parameter is 0%, i.e., all items are created as non-transparent. Transparency may be increased as required. 100% Transparency means that the object is invisible.

Upon setting the display parameters, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.



To apply the image generation changes after enabling or disabling **Use hardware acceleration**, KOMPAS-3D shall be restarted.

9.1.11.6. Changing Orientation

Adjustment of orientation is performed in the **Orientation change** called by the **Service — Parameters... — System — Model Editor — Orientation change** command. This dialog permits setting display parameters for reorientating (scaling) and after model orientation has been changed, as well as also selecting a predefined orientation when editing a sketch while performing operations.

Tab. 9.1.52. Control elements of changing orientation setup dialog

Element	Setting description
Smoothness	The option allows to enable the display of intermediate frames between the frames showing the model in its initial and final position (scale). Due to this, the model orientation (scale) transition appears smooth rather than abrupt. When the Smoothness option is enabled, you can adjust the number of intermediate frames by moving the slider between the positions Less and More . As the number of frames is decreased, the model display transitions become less smooth but faster whereas increasing the number of display transition produce the opposite effect.
Center Image	The option serves to enable centering of the model* image when changing its orientation, including automatic reorientation when creating sketches and operations (the reorientation takes place if the options When editing sketch and At Operation creation , respectively are enabled).
Maintain current window scale	The option serves to enable saving the scale set in the window model upon changing model orientation. If the option is disabled, then, upon changing the orientation, including automatic reorientation when creating sketches and operations, the model will display scale changes (see. table 9.1.53).
When editing sketch	The option allows to enable automatic Normal To orientation when creating a new sketch. When exiting the sketching mode, the model returns to the previous orientation. If the model orientation will change during working with the sketch, then this orientation will be restored upon its future editing. If the option When editing a sketch is turned off, then the model orientation does not change when creating or editing a sketch.
At Operation creation	The option allows to enable automatic setting of the selected orientation when creating a new geometry-producing feature, sheet body or a new surface. To specify an orientation, expand the list and select the desired item. Upon exiting an operation, the current model orientation will be saved. Editing operations does not change the model orientation /

- * **Centering** means aligning of the document window center with the center of the model circumscribing block projection. A window center is determined without regard to the area occupied by the Design Tree.

Tab. 9.1.53. Rules of model scaling at orientation changing

Method of changing orientation	Scale fitting rules
Selection of required orientation manually Automatic setup to selected orientation when creating operation	The scale will be selected to fit in the model window its outline box— a conditional block whose faces are parallel to coordinate planes and pass through the points of model items which are most distant from each other.
Automatic setup to orienting Normal to... orientation when creating a sketch	When creating a sketch on a projection plane, the scale is set to 1.0. When creating a sketch on a part face or on an auxiliary plane, the scale is picked so as to have the face or plane fully fit in the model window. Dimensions of the auxiliary plane are defined by the dimensions of the rectangle that represents it on the screen.

Upon setting the display parameters, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.11.7. Perspective Projection

Setting Up Options of Perspective Projection (degree of the added perspective distortion) for all windows is performed in the **Perspective Projection** dialog called by the **Service—Parameters... — System— Model Editor— Perspective Projection** command. When adjusting parameters of the perspective projection for the current window, the same dialog is used.

Dialog contains **Distance in Model Size** field. The value in this field shows, by how many times the distance from the model to the image plane is greater than the maximum model dimension. In other words, the model is shown on the screen in such a way that could be obtained by an optical instrument located at the specified distance from the model. The lesser the specified distance is, the more noticeable will be the image distortion on the bar.

To finish the setup, click the **OK** button. To leave the dialog without saving change, click the **Cancel** button.

9.1.11.8. Editing

Setup of selected and specified object colors is performed in the **Model editing** dialog called by the **Service — Parameters... — System — Model Editor — Edit** command.

The dialog serves to define the colors

- ▼ of model objects when selected and highlighted,
- ▼ and the component colors when
- ▼ components that make up the “environment” when one of the components is edited in the context of the model it is contained by (see Section 2.11.6.2.1 on p. 724),
- ▼ ghosts.

Tab. 9.1.54. Control elements of colors for selected and specified objects setup dialog

Element	Setting description
Table of Object Colors	The first table column contains colors and the second one contains names of the object types using these colors. To change a color, click the mouse on its cell, then press the button with triangle that appeared in the right part of the cell and choose the desired color in the standard color selection dialog.
Highlighting — Selected Object	Color of the selected objects. You often need to select objects before running a command. For example, to construct a form-generating 3D element, select its sketch; to make a copy of a graphic object— a source graphic object.
Highlighting — Selected Object1/ Selected Object2/ Selected Object3	Colors of objects (or object groups), selected when carrying out an operation. Objects must be selected in order to perform certain commands. For example, to construct a swept feature specify its path; to construct a parallel line— other line segment or straight line. Many-colored highlighting allows to distinguish between objects when specified since some operations use not one but several objects performing different functions. For instance, when a pattern is created on a parallelogram grid, all source objects will be highlighted in the color of the first object, the object chosen as the first axis will be highlighted in the color of the second object and the object chosen as the second axis will be highlighted in the color of the third object. Color specified for the first selected object is also used to highlight objects during dynamic search (dynamic highlighting) * if the Inversion during Dynamic Highlighting option is disabled.
Context editing— Passive Components	Color of Passive Components. Passive components are components that make up the “environment” when a component is edited in the context of the model that contains components. If the Use inherent component colors option is enabled, then the defined color will be ignored.
Ghost — Faces	The color of faces of a ghost displayed on the screen when creating or editing 3D element. Since the ghost is displayed semitransparent, its view also depends on transparency setting— lattice or realistic (see Section 9.1.11.5 on p. 1728).

Tab. 9.1.54. Control elements of colors for selected and specified objects setup dialog

Element	Setting description
Ghost — Captions	The color of captions within the feature's ghost, e.g. the numbers of lines when creating a surface by grid of surfaces.
Dimensions	Color of dimensions in sketches and operations.
Paint Faces when Highlighted	Enable this option to fill object faces with the color selected for selection (highlighting) on their highlighting (selection). If disabled, color changes are only applied to edges on selection (highlighting). It shall be noted that faces are not filled at dynamic highlighting— color changes are applied only to the edges.
Invert Colors When Highlighting	The option allows to enable inverted dynamic highlighting of edges instead of highlighting with a constant color defined fir the first object. It is recommended to enable this option while working with complex assemblies— this helps speed up dynamic targeting.
Inherent Colors of Components in Context Edit Mode	Activating the option means that component colors do not change in the context editing.
If requested, use OpenGL search	Enabling this option means that the dynamic search is performed on the basis of OpenGL objects. In some cases, this allows to optimize the dynamic search process.

* Dynamic search is described in Section 2.1.4.1.2 on p. 131.

Upon setting the desired colors of selected and specified objects, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.11.9. LCS

Selection of default methods of work with LCS (Local Coordinate Systems, see Section 2.8.3.2 on p. 596) is performed in the **LCS** dialog. It appears on the screen after calling the **Services—Options... — System — Model Editor — LCS** command. Description of options contained in dialog is represented in Table9.1.55. By default, the options are enabled.

Tab. 9.1.55. Control elements of the LCS setup dialog

Element	Setting description
When creating an LCS to assign it as the current CS	<ul style="list-style-type: none"> ▼ If this option is enabled a new LCS is automatically assigned with the “current” status. ▼ If this option is disabled, after creation of a new LCS the previous LCS will remain the current one. <p>Irrespective of this option status, the user can make any coordinate system available in the model the current system (See Section 2.8.3.3 on p. 597).</p>
Create LCS only in the absolute coordinate system	<ul style="list-style-type: none"> ▼ If this option is enabled, the settings of all new LCS being created shall be defined with respect to the absolute coordinate system of the model (see Section 2.8.3.1 on p. 595). ▼ If this option is disabled, parameters of a new LCS are specified relating to the coordinate system which is the current one at the moment. Thus, an LCS hierarchy may be formed in the model.

Upon finishing the settings, click the **OK** button. To close the dialog without setup changes, press the **Cancel** button.

9.1.11.10. Versions

Creation of objects and variables in dependent versions of models are set up in the **Versions** dialog. It appears on the screen after calling the **Services— Options... — System — Model Editor — Versions** command). Description of options contained in dialog is represented in Table 9.1.56. By default, the options are enabled.

Tab. 9.1.56. Controls of the setup dialog for creation of objects and variables

Element	Setting description
Create object in dependent versions	If the option is enabled when creating a new object in a source version, the identical object will be created in all of its dependent versions. If the option is disabled, the new object will not be created in dependent versions.
Create variable in dependent versions	If the option is enabled when creating a new user variable in a source version, the identical user variable will be created in all of its dependent versions. If the option is disabled, the new user variable will not be created in dependent versions.

Upon finishing the settings, click the **OK** button. To close the dialog without setup changes, press the **Cancel** button.

9.1.11.11. Thread Parameters

The list of standard thread tables is set up and they are connected in the **Thread parameters** dialog. It appears on the screen after calling the **Services— Options... — System — Model editor — Thread parameters**.

The **Thread standards** table includes a default list of files with parameters of a standard thread. The **File name** column shows the file path and the **Thread type column** — thread type and name of the standard.

You can add parameters of standard threads to the list of files. Click **Add** to enter a new file to the table. After it is clicked, the standard **Choose file for opening** dialog appears on the screen.

Select the required file (should have the *xml* extension) and click **Open**. A line for the selected file will appear in the **Thread standards** table.

Enable the option to the left of the file name to attach a file. During hole construction, only those views of standard threads are available that are connected to the system.

If you want to exclude any thread view in the table from construction, disable the option to the left of the file name.

Use the arrow button **Move up/down** to move the line with the file name and thread type in the table.

To delete a file with thread parameters from the list, select the line with its name and click **Delete**. Upon finishing the settings, click the **OK** button. To close the dialog without setup changes, press the **Cancel** button.

9.1.11.12. Dimensions and designations

In the **Display of Dimensions and Symbols** dialog you can configure whether and how the dimensions and process symbols are displayed in the model. It appears on the screen after calling the **Services— Options... — System — Model Editor — Dimensions and Notations** command.

Controls contained in the dialog are described in Table 9.1.57.

Tab. 9.1.57. Control elements of the dimensions and symbols display setup dialog

Element	Setting description
Optimize Display	<p>This option controls display of dimension text and texts within conventional signs.</p> <ul style="list-style-type: none"> ▼ If the option is enabled, you can choose position of dimension texts (see below). The dimension text font size and the symbol text font size will not change when model display scale is changed. Settings of sketch dimensions (arrow length, jut of extension lines beyond a dimension line, etc) also remain unchanged when scaling. ▼ If the option is disabled, dimension text appears in the planes for creating dimensions. Whenever model display scale is changed, the dimension text font size and the symbol text font size will change accordingly. <p>Symbol text will be displayed in the symbol creation plane regardless of the Optimize display state.</p>
Place dimension texts	<p>A group of controls for choosing position of dimension texts:</p> <ul style="list-style-type: none"> ▼ In the dimension plane, ▼ In the screen plane horizontally, ▼ Within the screen plane.
Show	<p>A group of options that controls displaying dimensions of sketches and operations on the screen (see Section 2.9.2.2 on p. 632) and connecting lines.</p> <ul style="list-style-type: none"> ▼ Enable the Dimensions of sketches and operations option so that dimensions of sketches and operations were displayed on the screen during creation or editing. ▼ Enable the Connecting lines option so that connecting lines of assigned dimensions (for example, projection lines or lines that conditionally display an object with dimensions) are displayed on the screen.



Please note that optimization of dimensions and symbols display does not affect printed model images. The model is printed as it is shown on the screen with the **Optimize display** option turned off.

To finish setting up dimension and designation display in model, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.11.13. Size

Setup of model size is performed using the **Model size** dialog which is called by the **Service — Parameters... — System — Model Editor — Size** command.

Control elements of the dialog allow selecting types of model objects which must be taken into account when creating its circumscribing block— a conditional block whose faces are parallel to coordinate planes and pass through the points of model items which are most distant from each other.

A 3D box is used, for example:

- ▼ to execute the **Show All** command (both in the 3D constructional mode and in the sketch mode)— the display is scaled so that the projection of the model 3D box on the screen plane fits in the window,
- ▼ and to center an image after you have changed model orientation, the center of the 3D box projection aligns with the center of the model window.

To take account of the required type of objects when creating a 3D box of the model, enable the option to the left of the type's name in the dialog. When the option is disabled, object type will be discarded.



It is impossible to disable all options— the last enabled option will be unavailable for disabling.

Enable the **Respect Incorporate Hidden Objects** option to involve hidden objects of the specified types in determining model size.

Upon finishing settings, click the OK button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.11.14. Zones

The document level on which objects are selected at selecting with the help of a zone is set up in the **Zones** dialog called by the **Service — Parameters... — System — Model Editor — Zones** command.

When the **Select objects with a zone only on the level of the current document** option is enabled, objects are only selected on the current document's level. When the option is disabled, all objects within the assembly can be selected.

To finish the setup, click the **OK** button. To leave the dialog without saving change, click the **Cancel** button.

9.1.11.15. MP

If model mass and center of gravity are not specified by the User and are determined by calculations, then, on completion of model editing which may lead to its mass and position changing, their current values are deleted from the model. To obtain new values of mass and center of gravity coordinates, the MP should be recalculated. This may be done manually (by using the **Model Mass Properties** command, see section 2.15.2.4 on p. 817), or automatically.

By default, manual MP calculation is done during model saving. You can additionally enable automatic MP recalculation during reconstruction of the model or disable automatic MP recalculation. This customization is made in the **MP** dialog activated by the **Service — Settings... — System — Model Editor — MP** command.

You can enable one of the dialog options: **Recalculate MP When Saving** or **Recalculate MP When Rebuilding**, or both options.

Results of automatic MP recalculation are saved in a model file. When a model is inserted into another model, they are transferred to this model and used to calculate its mass properties.

If neither option is enabled, MP are not recalculated automatically.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.11.16. Simplifications

9.1.11.16.1. Model

Model display parameters in the simplified display mode (see Section 2.1.3.6 on p. 129) are set in the **Simplified Model Display** dialog. It appears on the screen after calling the **Services—Options... — System — Model Editor — Simplifications** command.

Tab. 9.1.58. Control elements of the simplified model display setup dialog

Element	Setting description
Level of Detail	Control element serves to set the level of component detalization and bodies of the model when working in the simplified display mode. To change the level of detail, move the slider between the positions More (slower) and Less (faster) . The smaller is the level of detalization, the larger components or bodies are turned into rectangular boxes and the faster are the changes of the model's position (scale).
Use Auxiliary Mode for Standard Components	Option serves to call the special mode of standard components simplifying (for example, the models inserted from the Standard Item Library). When the option is disabled, standard components will be simplified in general mode, i.e. are subject to the level of detail setup for components.
Simplify	Enable this option to have standard components with the simplified display mode displayed as rectangular boxes irrespective of the set level of detalization. The option is available when Use Auxiliary Mode for Standard Components option is enabled.
Hide	Enable this option to have the standard components hidden when working in the simplified display mode. The option is available when Use Auxiliary Mode for Standard Components option is enabled.

Tab. 9.1.58. Control elements of the simplified model display setup dialog

Element	Setting description
Delay in object search, ms	The option allows to turn on the delay in dynamic object search when the simplified display mode is enabled. Enter delay value (in milliseconds) in the field to the right of the option. This is the period of time for the cursor to stay on an object prior to starting dynamic search. So, if the option is enabled, a model object will not be highlighted immediately but only after the cursor stops on it. But if the option is disabled, the system will try to find and highlight an object immediately. It is not always possible when you work with large assemblies (the dynamic search can lag behind the cursor movement, i.e. the objects already passed by the cursor can be highlighted). Therefore, we recommend to enable the delay in search for such models.

To finish setting up simplified display, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.11.16.2. Other

You can turn on additional methods of simplification means that would help to even more to speed up working in the simplified model display mode (see Section 2.1.3.6 on p. 129). For this purpose, the **Other parameters of image simplification** dialog is used which is called by the following command: **Service — Parameters... — System — Model Editor — Simplifications — Other** command.

Tab. 9.1.59. Control elements of the other simplified display settings dialog

Element	Setting description
When Image is Modified	<p>The group of options makes it possible to enable or disable additional simplifications that will be applied in the simplified display mode when the image is modified later.</p> <ul style="list-style-type: none"> ▼ Moving and rotating a model with mouse or by the commands Shift and Rotate, ▼ reorienting and rescaling the model (if the smooth transition is enabled, see Section 9.1.11.6 on p. 1731), ▼ moving and rotating components. <p>A static image does not depend on the status of this option.</p>
Quick Lines Displaying	Option allows to use the quick display of model edges. If this option is enabled, the command Quick Lines Displaying* is automatically activated when you modify an image.

Tab. 9.1.59. Control elements of the other simplified display settings dialog

Element	Setting description
Hide construction axes, planes etc.	Option allows to control displaying of auxiliary and some other objects of the model. The following commands will be activated when the image is modified, if the option is enabled: <ul style="list-style-type: none"> ▼ Hide — Origins of Coordinate Axes, ▼ Hide — Construction Planes, ▼ Hide — Construction Axes, ▼ Hide — Sketches.
Hide Surfaces	Option allows to control the display of surfaces present in the model window. The Hide— Surfaces command will be activated when the image is changing, modified, if the option is enabled. When the option is disabled the surfaces are still displayed and are simplified according to the set level of detalization.
Hide 3D Lines	Option serves to control displaying 3D lines— polylines, splines and spirals. The Hide— 3D Lines command will be activated when the image is modified, if the option is enabled.
Disable “Half-tone Wireframe” mode	Option serves to control the use of aforementioned type of displaying a model. If this option is enabled the components or solids bodies that were not turned into boxes will not use the Dimmed Wireframe display mode (if it was enabled) when the image is modified.
Disable Outlines	Option allows to disable outlines ^{**} of components or bodies.
Hide Grid	Option allows to disable grid when scaling sketch.
Disable highlighting of the selected objects.	Option allows to disable the highlighting of selected objects (except faces) when the image is modified. Selected objects will be highlighted if the option is turned off. The presence or absence of Selected 3D faces do not affect the speed of display transitions.
Lattice Transparency	Option allows to enable displaying of transparent objects (if any) as a pixel grid.
Hide Control Points, Hide Reference Threads Image, Hide Dimensions, Hide Reference Marks	Options serve to disable drawing of control and connection points, threads images, sizes and notations.

Tab. 9.1.59. Control elements of the other simplified display settings dialog

Element	Setting description
Display hidden objects in form of points	<p>Option allows to enable displaying of points in position of components (or bodies) which became hidden due to small scale. Objects which became hidden can be displayed as points only in simplified displaying mode.</p> <p>If the option is turned off, small components and bodies can totally “disappear” under a certain reduction scales</p> <p>Standard components also can become hidden due to enabling option Hide in the setup dialog of model simplification (see Section9.1.11.16.1).</p>

* Command controls the same-name mode. When the mode of quick lines displaying is enabled, drawing of lines depicting the model with hidden lines removed, with thin hidden lines or in the wire frame mode is speeded up.

Speeding up the model drawing is achieved by applying the so-called graphical method of calculating model lines. When the quick display mode is disabled, these calculations are run analytically. A mathematical calculation gives a slightly more accurate result, but that takes much more time.

** Contour line is a line in whose points the face's normal is perpendicular to the sight direction.

To finish setting up simplified display, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.1.12. Reports

9.1.12.1. Report style list setup

Styles from style library are connected in the **Setup of report style list** which is called by the **Service — Parameters... — System — Reports — Setup of report style list** command.



The dialog can also be called when the report is being created by choosing the **Other...** line in the list of styles in the Property bar.

The dialog allows to specify previously created style libraries (the *.lrf files) and select styles from them to use in the reports.

Tab. 9.1.60. Control elements of the report style list customization dialog

Element	Setting description
The Style Libraries tab	<p>Controls of the tab serve to attach styles libraries</p> <p>The list of libraries represented as a table is displayed on the tab. Every table row contains library file name and its path. Add and Remove buttons allow to edit the list contents.</p> <p>To attach desired library, enable the option next to its name.</p> <p>Styles of the connected libraries can be selected in the Styles tab.</p>

Tab. 9.1.60. Control elements of the report style list customization dialog

Element	Setting description
Add	The button allows to add a library to the list. After clicking it, the standard File Selection dialog will appear.
Delete	The button allows to delete a selected library from the list.
 Move Down/Up	Buttons allow to move libraries within the list. To move a library up, select its name and press the Move Up button, to move it down — the Move Down button. The selected library will move by one position in the specified direction.
 Tab Styles	Tab controls serve to connect the styles used in the report. The tab shows a style list from the libraries connected in the Style Libraries tab. The list is represented as a table. Every table row contains style name and path to a library file containing this style. To attach desired style, enable the option next to its name.
All	Option allows you to attach/detach all styles contained in the list.
Preview	The button serves to view selected style parameters. After you press it, the style parameters setup dialog will pop up in the view mode.

Objects located in the dialog can be sorted by any of the columns. To select the column to be sorted, click the mouse on its header. A triangle will appear next to the header. It shows the direction of sorting. If the vertex of the triangle is faced down, then the sorting of the list is done by descending the values in this column, and if up— by ascending.

Connect a report style library on the **Style Libraries** tab and a style on the **Styles** tab. The sequence of steps is described in Section 5.2.2.2.8 on p. 1415.

9.1.12.2. Scroll Bars

Controlling the scroll bar display and the image auto shift in the window is done in the **Scroll Bar Setup** dialog called with the **Service— Parameters... — System— Reports— Scroll Bars** command. Working with the dialog is described in Section 9.1.7.3 on p. 1706.

9.1.12.3. Table Positioning

9.1.12.3.1. Sheet Format

Setup of the report's page format is performed using the **Page format** dialog which is called by the **Service — Parameters... — System — Reports — Table positioning — Page format command**.

The dialog serves to select a sheet size from a standard set or to define arbitrary sheet dimensions.

The same dialog is used when setting the format of text and graphic document pages, as well as additional sheets of BOM.

Tab. 9.1.61. Control elements of the sheet format setup dialog.

Element	Setting description
Standard	Option allows to specify dimensions of document sheet according to one of the standard formats. If this option is enabled, the required format selections become available and its parameters can be set.
Designation	The drop-down list allows you to select the desired format.
Multiplicity	The field allows to type or set by the increment button the sheet size in multiples of the standard paper size.
Orientation	Group of options allows to choose a document sheet orientation— Horizontal or Vertical .
Custom	The option serves to define arbitrary sheet dimensions. If the option is enabled, fields for dimensions setting become available.
Width, mm Height, mm	Fields allow to enter arbitrary document sheet dimension values.
Apply to section	The list is used to select the text document section that settings made in the dialog will be applied to. The list is only present in the dialog at setup of the current text document and available if the document has two or more sections.

Upon finishing with page format settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.1.12.3.2. Sheet fields

You can preset dimensions of the printing area frame on page. For this purpose, the **Page margins** dialog is used which is called by the following command: **Service — Parameters... — System — Reports — Table positioning — Page margins**.

Enter or set by the increment button the offset values (in millimeters). Offsets are measured from the sheet size borders

To finish the setup, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2. New and current document parameters

9.2.1. General information about new and current document

Setting of parameters of New and Current document is made on dialog called with, **Service – Parameters...** command

To perform New document setup you shall use **New document** tab of this dialog (Fig. 9.2.1). In the left part of the tab there is the object setup list represented as a “tree” They are grouped by sections according to their types. After selecting an item in the left side, control elements will appear in the right side of the tab.

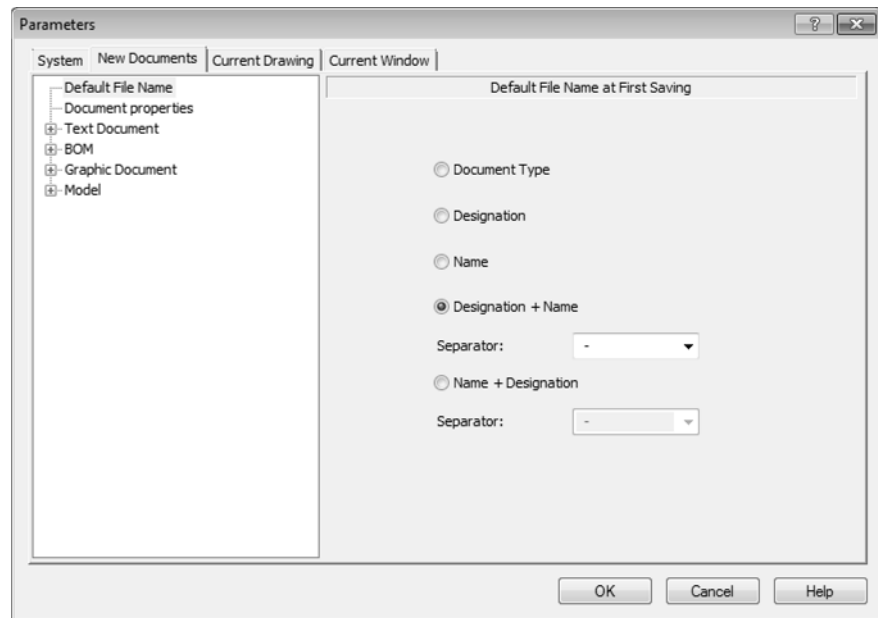


Fig. 9.2.1. Default filename setup dialog

After closing a dialog with **OK** the information about new documents is stored in the file **.prj* (see Section 9.5.2 on p. 1881) and used in creation of new documents by default.

After the document has been created, its configuration is saved in it. Changes of configuration of the existing document is carried out in a special tab of the setup dialog. Depending on a type document, it has one of following names:

- ▼ **Current drawing** (Fig. 9.2.2),
- ▼ **Current fragment**,
- ▼ **Current assembly**,
- ▼ **Current part**,
- ▼ **Current text document**,

▼ Current BOM.

When working with a sketch, the dialog contains **Current sketch** tab.

As well as the tab for performing new documents setup, tab for performing setup of current has object setup tab in its left part and control elements in its right part.

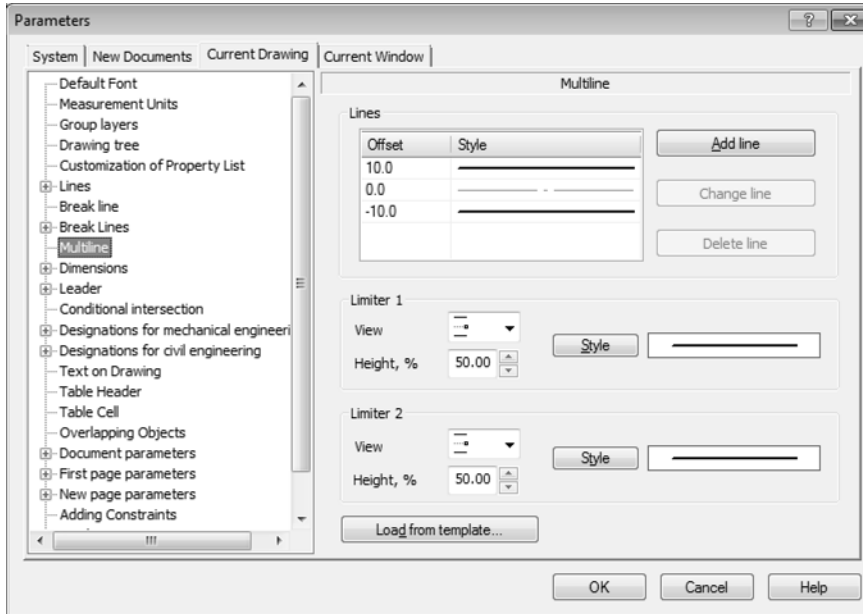


Fig. 9.2.2. Multiline setup dialog in current drawing

Please pay attention to the following conventions and simplifications established in this Section:

- ▼ Further not all setup dialog, and only set control elements will be implied by “dialog” for adjustment of this or that object.
1. Calling to this set of control elements will be described in the following form: **Service — Parameters... — Name of the Setup Dialog Tab — Section Name** (group of setup objects) — **Subsection Name** (subgroup of setup objects). For instance, if the text says: «...This setting is performed in the **Units of measurement of length**, invoked by **Service— Options— New documents— Graphic Document— Units of measurement...**», command and this means that in order to get access to units of measurement setup, it means that you have to perform this sequence of operations. Select the item **Service** in the Main menu.
 2. Select the command **Options...** in the list of commands of the menu **Service** that appears.
 3. In the dialog that appears, activate the tab **New documents**.
 4. In the list of setup objects, expand **Graphic document**.
 5. Select the following section: **Measurement units**.
The right part of the tab will display control elements with a general title **Units of measurement of length** used to select length measurement units in the drawing .

9.2.1.1. Types of objects' parameters

According to their effect on **objects of current document** all adjustable parameters can be subdivided into two groups each containing two subgroups. In the setup dialogs subgroup's number to which this or that parameter belongs is specified.

1. **Document parameters** are parameters that cannot be changed while creating objects.
 - 1.1. The parameters that define objects' external features, which must be the same for all the objects of a specific type that exist in the same document. Changing these parameters affect both the existing objects, and the newly created ones.
 - 1.2. Parameters that affect the availability of certain options while creating objects. Changing these parameters does not affect the existing objects, but does affect newly created objects.
2. **Object properties** are parameters that can be changed while creating object.
 - 2.1. Parameters that define the default values of the corresponding object properties. Changing them leads to changes in the default settings of new objects. The existing objects do not change. To change the parameter values (in this subgroup) of the existing objects, you will need to edit these objects.
 - 2.2. These parameters are similar to those in the previous subgroup with one major difference: after changing these parameters, the new values are transferred to those existing objects which were created with the default values of the respective parameters. All these objects will be redrawn. Objects created or edited with manually changed parameters will be left unchanged. To change these parameters in some objects, you will need to edit these objects.

9.2.2. File name by default

You can configure the default filename that will be suggested when you save a file for a first time. Settings are changed in the dialog **Default File Name at First Save**, invoked with the **Service — Parameters... — New documents — Default file name**

Tab. 9.2.1. Default filename setup dialog control elements

Element	Setting description
Document Type	Choose this option so that the savable document type name is offered as a file name. For instance, when you save a drawing, you will be offered the name "Drawing.cdw", and for an assembly — "Assembly.a3d".

Tab. 9.2.1. Default filename setup dialog control elements

Element	Setting description
Designation	Select this option so that the savable document designation name is offered as a file name. When you save a drawing, BOM or text document, there is used text entered into the appropriate cell of the title block is used, and when you save a part or assembly, text entered into the Designation field of the Property bar when model properties are customized. If designation is not specified, the document type name is offered as a document name.
Name	Select this option so that the savable document name is offered as a file name. When you save a drawing, BOM or text document, there is used text entered into the appropriate cell of the title block is used, and when you save a part or assembly, text entered into the Name field of the Property bar when model properties are customized. If no name is specified, the document type name will be suggested as the filename.
Designation + Name, Name + Designation,	A file name may consist of two parts — designation and name — with a separator between them. Select the required name type, and enter or select from the list a separator between name parts. If either name part is not specified, the other part is used without a separator. If neither name part is specified, the name of the document type name is offered as a document name.



Hence, when saving fragments you are offered the name “Fragment.frw” regardless of the setting.

Upon finishing with display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.3. Document properties

Toggle on or off filling document properties, as well as setting up default document information is carried out in **Common Properties** dialog. It appears on the screen after calling the **Services— Options... — New Documents — Document Properties**.

Tab. 9.2.2. Dialog for Document General Property Settings control elements

Element	Setting description
Offer Filling Document Properties	The option controlling the automatic opening of the document information dialog when KOMPAS documents are first saved.
General Document Information	This is a group of fields to edit general document information. By default, the author and company information match those specified when installing KOMPAS-3D (in the User Information dialog). Data entered in this dialog will be by default displayed in the respective fields of the properties dialog. If necessary, you can change them for a particular document.

Upon finishing with general document settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.4. Text document

9.2.4.1. Font by default

Selection of a font to use by default for all text documents (new and current) is carried out with **Default Font** dialog, invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Text document — Default font**,
- ▼ for the current document — **Service — Parameters... — Current text document — Default font**.

A similar dialog is used when setting up the default font:

- ▼ in all texts of the new or current graphic documents (except for the title block),
- ▼ in dimensions and designations of new and current models.

drop-down list **Font** enables selection of a font, which will be applied by default for a text and tables in documents.

Example field serves to display a sample of selected font.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

Selected font is used as default font for all document objects (text of document, table heading, table cell etc.). In order to select another font for an object (for example, for the text in surface finish symbols, or for table cells), select corresponding item in the left part of parameters setting up dialog. Control elements for text parameters setup will appear on the right side of the dialog box. Enable **Different from Default Font** option and select another font. Further changes to the default font will not affect the local font selected for the object. Instead, upon disabling the option **Different from Default Font**, the default font will be used again for objects of the given type.



For some objects **Different from Default Font** option is located in **Font Options** dialog, invoked by **Font...** button.

When editing settings of current document, editings are saved in the document itself and document objects are automatically updated.

If special font was selected in document window when creating or editing an object, then default font changes do not affect it.



Vector fonts including fonts GOST type A (plotter), GOST type B (plotter), Symbol type A (plotter), Symbol type B (plotter) supplied with KOMPAS-3D are not supported in the mode of work with 3D model and graphic documents if OpenGL is used for their displaying on the screen (see. Chapter 9.1.7.17 on p. 1719).

9.2.4.2. Sheet Parameters

9.2.4.2.1. Format

You can select text document (new and current) sheet size from a standard set or define arbitrary sheet dimensions. To do this, use the **Sheet Format** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Text document — Sheet parameters — Format**,
- ▼ for the current document — **Service — Parameters... — Current Text Document — Sheet Parameters — Format**.

Working with the dialog is described in Section 9.1.12.3.1 on p. 1743.

9.2.4.2.2. Template

Sheet layouts for text documents (new and current) are carried out in **Layout** dialog invoked by command:



- ▼ for new documents — **Service — Parameters... — New documents — Text document — Sheet parameters — Layout**,
- ▼ for the current document — **Service — Parameters... — Current Text Document — Sheet Parameters — Layout**.

The similar dialog is used in selection of sheet layouts for graphic and text documents, as well as selection of the BOM style (in the last case, dialog is called **Style**).

For text documents layout of the first sheet and subsequent sheets is selected from dialog (even and odd separately), for graphic documents - of the first sheet or all again added sheets, for specifications — style of design of specifications.

Document templates and BOM styles are stored in special system libraries: files with extension *.lyt*. Controls of the dialog make it possible to select the file of needed library and template (style) from it.

Tab. 9.2.3. Control elements for layout selection dialog or style

Element	Setting description
Library	The field serves to display full library file name <i>*.lyt</i> .
 Show Library	The button allows to select desired template library file (styles) in the standard file selection dialog. After you select a library, a dialog, showing a list of layouts contained in it is displayed (styles). Specify the required template (style). Its name will appear in the field Name , and full name of selected library is in the Library field.
Name	The field serves to display layout name, assigned to a sheet (style, assigned to BOM). Layout (style), embedded to the current, i.e. already existing, document can differ from layout in <i>*.lyt</i> library, corresponding to it (differences may appear, for example, after library editing). In this case, before layout (style) name, «asterisk» (*) is displayed.
 Select	The button serves to select another layout (style) from current library of layouts (styles). After you push button, a dialog will appear on the screen, showing a list of layouts (styles), contained in current library. Specify the required template (style) and push OK button.
Clear	The button serves to detach the current library of layouts (styles). In this case, Library and Name fields are cleared. The document with no layout has no frame and title. BOM does not exist without style, that is why there is no Clear button in BOM style selection dialog.
Re-read	Allows to update layout (style), provided that it differs from those in the library. After pressing this button, layout from the library will be assigned to a sheet, (style from a library will be assigned to BOM), «asterisk» before the name of the layout (style) will disappear. Re-read button present in the dialog, if it is invoked for current document. It is available, if document layout (style) differs from corresponding layout (style) in <i>*.lyt</i> library.
Apply to section	The list is used to select the text document section that settings made in the dialog will be applied to. The list is only present in the dialog at setup of the current document and available if the document has two or more sections.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.4.3. Document text

Setup of paragraph parameters for text documents (new and current) are carried out in **Text document** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Text document — Document text**,
- ▼ for the current document — **Service — Parameters... — Current Text Document — Document text**.

The similar dialog is used when setting up:

- ▼ table title in text or graphic document,
- ▼ table cells in text or graphic document,
- ▼ report table header,
- ▼ report table cells,
- ▼ text on drawing,
- ▼ drawing technical specifications text.

Tab. 9.2.4. Control elements of the document text setup dialog

Element	Setting description
Line spacing, mm	The field to enter the distance between lines in paragraph. Measurement of line spacing allows to keep proportions of distances between lines typed using a font with various symbol heights (for instance, to introduce into the text some footnotes, comments or remarks typed in a smaller font compared to the main text).
First line Indent, mm	The field to enter paragraph first line offset. Indent is measured from text boundary. When creating a new paragraph, the cursor is automatically set to a position with preset indent.
Indentations	The fields for entering distances between left and right input field boundaries and corresponding text borders.
Spacing	The field to enter the distance between two contiguous paragraphs. The given distance is formed by two values— the interval after the first paragraph and the interval before the second paragraph. Setting of intervals allows to separate paragraphs in a text document for their optimal perception while reading, as well as to draw attention to especially important information.
Alignment	Options serve to define desired paragraph alignment method (left, right, centered, justified).
Font...	A button to call the font parameter setup dialog.
Tabs...	A button to call the dialog to set up tabulation parameters.

Upon finishing paragraph parameter setting up, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.4.4. Table header

Setup of table header for text documents (new and current) are carried out in dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Text document — Table heading**,
- ▼ for the current document — **Service — Parameters... — Current Text Document — Table heading**.

Work with dialog is described in Section 9.2.4.3 on p. 1752.

9.2.4.5. Table cell

Setup of table cell for text documents (new and current) are carried out in dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Text document — Table cell**,
- ▼ for the current document — **Service — Parameters... — Current Text Document — Table heading**.

Work with dialog is described in Section 9.2.4.3 on p. 1752.

9.2.4.6. Sheet numeration

Setting up of sheet numbering of text document is done in the **Sheet Numbering** dialog invoked with **Service — Parameters — Current text document — Sheet numbering** command.

Numbering setup serves to define the rules of filling in the title block entries *Number of Sheets* and *Sheet number*.

Tab. 9.2.5. Control elements of the sheet numbering setup dialog

Element	Setting description
Automatic Sheet Numbering	The option controls automatic sheet numbering. If checked, all sheets of a document are automatically numbered. If unchecked, the <i>Sheet number</i> field in the title block of the current document is not filled in. You can enter an arbitrary number for each sheet and an arbitrary total number of sheets.
Start numbering from	The field containing number of the first page— is the number from which automatic numbering will begin. By default, it is equal to one. You can type in or set by the counter button the desired value. The field is available when the Automatic page numbering option is enabled.

Tab. 9.2.5. Control elements of the sheet numbering setup dialog

Element	Setting description
Continue numbering	This option is used to continue the numbering from the previous section. Available when the Automatic sheet numbering option is enabled.
Number of Pages	A group of switches allowing to select the way of determining the number of sheets in a document.
Auto Determination	The option for enabling automatic counting the sheets of the current document and entering the <i>count</i> into the corresponding cell of the title block. If the Auto Determination option is enabled, manual data entry in the <i>Number of sheets</i> cell of the title block is impossible.
Predefined	The option serves to define arbitrary sheet dimensions. This may be needed, if the current document is part of another document and consists of a specified number of sheets The field to the right of the option allows to enter required value manually or set by the incrementer. Set value is entered into <i>Number of sheets</i> cell of the title block of each sheet in the current document. When checked, manual input to this entry will be impossible.
Exclude from calculation	This option is used to exclude a document section from numbering.
Apply to section	The list is used to select the document section that settings made in the dialog will be applied to. The list is present in the dialog if the document includes two or more sections.

Upon finishing sheet numbering settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.4.7. Report table parameters

9.2.4.7.1. Title

Table title text parameters are carried out in dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Text document — Report table parameters — Heading,**
- ▼ for the current document — **Service — Parameters... — Current text document — Report table parameters — Heading.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

Setup is used when placing a report in the text document.

9.2.4.7.2. Cell

Cell table text parameters are tuned in dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Text document — Report table parameters — Cell**,
- ▼ for the current document — **Service — Parameters... — Current text document — Report table parameters — Cell**.

Work with dialog is described in Section 9.2.4.3 on p. 1752.

Setup is used when placing a report in the text document.

9.2.5. BOM

9.2.5.1. Default settings

9.2.5.1.1. Style

BOM layouts for text documents (new and current) are carried out in **Style** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — BOM — Style**,
- ▼ for the current document — **Service — Parameters... — Current BOM — Style**.

Work with dialog is the same as described in Section 9.2.4.2.2 on p. 1750.

9.2.5.1.2. Sheet numbering

Setting up current BOM sheet numbering is done in the **Sheet Numbering** dialog called with **Service — Parameters...** command. — **Current BOM — Sheet Numbering**. A similar dialog is used to set up the drawing sheet numbering.

Tab. 9.2.6. Controls of the sheet numbering setup dialog

Element	Setting description
Automatic Sheet Numbering	The option controls automatic sheet numbering. If checked, all sheets of a document are automatically numbered. If unchecked, the <i>Sheet number</i> field in the title block of the current document is not filled in. You can enter an arbitrary number for each sheet and an arbitrary total number of sheets.
First Sheet Number	The field containing number of the first page— is the number from which automatic numbering will begin. By default, it is equal to one. You can type in or set by the counter button the desired value. The field is available when the Automatic page numbering option is enabled.
Number of Pages	A group of switches allowing to select the way of determining the number of sheets in a document.

Tab. 9.2.6. Controls of the sheet numbering setup dialog

Element	Setting description
Auto Determination	The option for enabling automatic counting the sheets of the current document and entering the <i>count</i> into the corresponding cell of the title block. Additional pages (if any) are included in the total number of sheets. If the Auto Determination option is enabled, manual data entry in the <i>Number of sheets</i> cell of the title block is impossible.
Predefined	The option serves to define arbitrary sheet dimensions. This may be needed, if the current document is part of another document and consists of a specified number of sheets The field to the right of the option allows to enter required value manually or set by the incrementer. Set value is entered into <i>Number of sheets</i> cell of the title block of each sheet in the current document. When checked, manual input to this entry will be impossible.

Upon finishing additional page settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.5.1.3. Additional pages

BOM additional sheets control (new and current) is carried out in **Additional sheets** dialog, invoked by command:



- ▼ for new documents — **Service — Parameters... — New documents — BOM — Additional sheets**,
- ▼ for the current document — **Service — Parameters... — Current BOM — Additional sheets**.

The dialog serves to create additional pages at the beginning and/or at the end of the document, to select designs, format and to set the order of document sequence. For example, you can use this dialog to create a title page and a revision page in a document.

Tab. 9.2.7. Controls of the document additional pages setup dialog

Element	Setting description
At start of a document, At the end of a document	Lists of names of additional sheet layouts. The number of additional sheets, as well as the sequence of sheets in a document is defined by the number and location in lists. You can add new sheets in a list, to delete them, to change their location and design.
Add	Allows to create an additional sheet in document. On pressing the button the sheet layout selection dialog will be displayed. The dialog to select template library and to specify desired layout. Selected layout appears in corresponding Additional sheets dialog list.

Tab. 9.2.7. Controls of the document additional pages setup dialog

Element	Setting description
Change	The button allows to modify additional sheet layout. The layout of this sheet shall be selected before in dialog list. On pressing the button the layout selection dialog will be displayed.
Format	The button allows to modify additional sheet format. The layout of this sheet shall be selected before in dialog list. Upon pressing the button, the dialog for sheet format settings will appear on the screen. You can select one of standard formats or enter custom lengths of sheet sides.
Delete	Allows to delete the selected additional sheet from the document. The layout of this sheet shall be selected before in dialog list.
 Move Up/Down	Order of additional sheet sequence in a document is defined by sequence of allocation of their layout in dialog lists. Buttons make it possible to change the sequence. In order to change position of additional sheet in a document, you shall select its layout in correspondent list and press Move up button (to move a sheet forward) or Move down (to move a sheet backward).
	

Upon finishing additional page settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

Working with additional BOM sheets is described in Section 6.2.6.9 on p. 1529.

9.2.5.1.4. Display of values

It allows to set mass values and display accuracy (quantity of digits after the comma) to use in the corresponding BOM columns, e.g. in the *Mass* column of sheer BOM or in the additional column named *Mass*. This setup is done in the **Display of values**, invoked by a command:

- ▼ for new BOMs — **Service — Parameters... — New documents — BOM — Default Settings — Display of values**,
- ▼ for a current BOM — **Service — Parameters... — Current BOM — Display of values**.

Select mass units and set decimal places in the mass value in the appeared dialog.

Upon finishing with settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

After adjustment of current BOM display of mass value in corresponding cells changes.

The set decimal places is used only for mass value displaying. That is the mass value is constantly stored in the cell with that accuracy, with which it was entered by the user or received from the model, but it is displayed in the cell rounded to the specified decimal place.

9.2.5.2. By assembly with versions

9.2.5.2.1. Setup option selection parameters

A BOM created on the basis of an assembly with versions, can have a different setup depending on the number of the assembly's versions.

The number of versions, with respect to which a setup variant is assigned to BOM, is set in the **Parameters of setup variant selection** dialog which is called by the **Service — Parameters... — New documents — BOM — By assembly with variants — Setup variant selection parameters**.

The **If number of versions is not more than** option controls the application of a special setup variant to BOM. The field on the right of the option makes it possible to set the number of versions with respect to which the setup variant to be applied is defined (available if the option is enabled).

If the option is enabled:

- ▼ for a BOM created in an assembly with the number of versions is not more than the number specified in this field, the parameters set in the **If the number of versions is not more than** section (see Section 9.2.5.2.3 on p. 1759),
- ▼ for a BOM created in the assembly with the number of versions is more than the number specified in this field, the parameters set in the **General variant section** (see Section 9.2.5.2.2).

When the option is disabled, a BOM with parameters specified in the **General variant section** is created for an assembly with any number of versions.

9.2.5.2.2. General variant

In this section, the variant of settings is customized which will be used for the BOM created on the basis of assembly the number of versions of which is not more than the number set in the **Setup variant selection parameters** (see Section 9.2.5.2.1).

Style

Selection of style for new BOMs created on the basis of an assembly with versions, is performed in the **Style** dialog which is called by the **Service — Parameters... — New documents — BOM — By assembly with versions — General variant — Style** command.

Work with dialog is the same as described in Section 9.2.4.2.2 on p. 1750.

Additional pages

Setting additional pages' parameters for new BOMs created on the basis of an assembly with versions, is performed in the **Additional pages** dialog which is called by the **Service — Parameters... — New documents — BOM — By assembly with versions — General variant — Style** command.

Working with the dialog is described in Section 9.2.4.3 on p. 1752.

Displaying values

Selection of the mass measurement unit and specifying decimal points in the value of mass for new BOMs created in an assembly with versions is performed in the **Display values** dialog

which is called by the **Service — Parameters... — New documents — BOM — By assembly with versions — General variant — Display values** command.

Working with the dialog is described in Section 9.2.5.1.4 on p. 1757.

9.2.5.2.3. When the number of versions is no more than the specified number

In this section, the variant of settings is customized which will be used for the BOM created on the basis of assembly the number of versions of which is not more than the number set in the **Setup variant selection parameters** (see Section 9.2.5.2.1 on p. 1758).

Style

Selection of style for BOMs created by an assembly with the number of versions (the number of versions does not exceed the set number), is performed in the **Style** dialog which is called by the **Service — Parameters... — New documents — BOM — By assembly with versions — When the number of versions is no more than the specified number** command.

Work with dialog is the same as described in Section 9.2.4.2.2 on p. 1750.

Additional pages

Setting additional pages' parameters for new BOMs created by an assembly with the number of versions not exceeding the specified one (the number of versions does not exceed the set number), is performed in the **Additional pages** dialog which is called by the **Service — Parameters... — New documents — BOM — By assembly with versions — When the number of versions is no more than the specified number — Additional pages** command.

Working with the dialog is described in Section 9.2.4.3 on p. 1752.

Displaying values

Selection of the mass measurement unit and specifying decimal points in the value of mass for new BOMs created in an assembly with the number of versions not exceeding the specified one is performed in the **Display values** dialog which is called by the **Service — Parameters... — New documents — BOM — By assembly with versions — When the number of versions is no more than the specified number — Display values** command.

Working with the dialog is described in Section 9.2.5.1.4 on p. 1757.

9.2.6. Graphic document

9.2.6.1. Default font

You can select a font to use by default for all texts of new and current graphic document (except for drawing title).

To do this, use the **Default font**, dialogue, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Default font**,

- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Default font.**

Working with the dialog is described in Section 9.2.4.1 on p. 1749.

9.2.6.2. Units

You can select length and mass units in graphical documents (new and current). It will be used to display commands parameters (for example, a length or a radius), coordinates, cursor step, automatically generated nominal dimension values and tolerances. This setup is done in the **Length values**, invoked by a command:

- ▼ for new documents — **Service — Parameters... — New Documents— Graphic Document— Measurement units.**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Measurement units.**

To select required unit, activate the respective option.

To finish the setup, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



The length units setting up does not cover the Window of Variables (see Section 7.1.1.2 on p. 1569). All linear values (e.g., linear dimensions) displayed in this window are converted to millimeters; all values entered to calculate such values are considered specified in millimeters.

9.2.6.3. Grouping of layers,

Grouping layers method in graphical documents (new and current) can be configured in the dialog **Layers grouping**. The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Grouping of layers,**
- ▼ for the current document — **Service — Parameters... — Current Drawing (Fragment) — Layers grouping.**

Tab. 9.2.8. Control elements of the layer grouping method setup dialog.

Element	Setting description
Group Layers	The option makes it possible to create and use group of layers and filters* .
Group Layer Properties	The option makes it possible to create and use layer properties group* . Inherit Parent Item Structure option is accessible when the option is activated.

Tab. 9.2.8. Control elements of the layer grouping method setup dialog.

Element	Setting description
Inherit Parent Item Structure	<p>Serves to automatically include layers of the parent object into the contents of Layer Properties group. The document, view or group where a new layer properties group is created, is considered parent. If this option is switched on, the creation of a properties group is performed as follows:</p> <ul style="list-style-type: none"> ▼ if the parent element is a properties group, then all layers of this group with their settings will be copied, ▼ if the current object is a view, then the new group will contain all layers of the view and their properties, ▼ if the parent object is the Document (root object of the Tree of sheets, views, layers), then the new group will contain all layers of all views.

* Description of groups of layers and layer properties group is described in Section 3.5.6.9.1 on p. 1192.



Modifying a grouping method in the document causes irreversible changes to the existing groups and filters.

If layer properties grouping mode was enabled in the document, then, after the layer grouping mode is enabled, the properties groups will become layer groups. The state of objects of a group layers will correspond to their current state in the document.

If layer grouping mode was enabled in the document, then, after the properties grouping mode is enabled, the layer groups will become properties groups. The state of objects of a group layers will correspond to their current state in the document. The group settings should be reset. Since filters cannot be used when layer properties grouping is enabled, all existing filters are automatically converted to layer properties groups. Their names will not change, i.e., will consist of the word "Filter" with previous sequential number.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.4. Drawing Tree

Set up of default contents of Drawing tree for new and current graphical documents is carried out in dialog **Display in the tree**, invoked by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Drawing's tree**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — Drawing's tree**.



Moreover, while you work with a drawing, drawing tree setup can be invoked by clicking the **Drawing Tree Contents** upper part of Tree window.

The dialog contains options corresponding to groups of objects in the Drawing tree. You can manage display of groups in Drawing tree activating and deactivating corresponding options.

You shall take into account that existence of empty groups in the Tree is impossible when adjusting Drawing tree display. Therefore, the sections in which the display is enabled will appear in the Tree only after the first relevant item type has been created. For instance, if a drawing does not have a single macro element, there will be no **Macro elements** group in the Tree.

After the Tree is customized, click the **OK** button. When you customize the current drawing, changes will be displayed in the Tree: groups will disappear or appear in it. When you customize new drawings, made changes that are made will take effect at their creation of these drawings.

To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.5. Properties list setup

You can customize necessary libraries connection or and properties display in graphic documents (both new and existing ones). A **Properties list setup** dialogue is used for this purpose; this dialogue is called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Property list setup**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Property list setup**.

Similar dialog is used for setting up properties list in model-documents (parts, simple and technological assemblies) and reports.



This dialog allows connecting property libraries and customize displaying of the required properties for new documents and reports. The dialog contains the **Property Libraries** and **Properties** pages.

The dialog allows only to customize the property displaying for the current document. In this case the dialog contains only one page: **Properties**.

Tab. 9.2.9. Control elements of the Properties list setup dialog

Element	Setting description
Property Libraries	This page allows to generate the list of libraries, the properties of which can be used in new documents and reports.
List of libraries	The list contains the names of files of property libraries selected by the user and file paths. An option is located to the left from the file name of each library. It allows to connect the property library. After it, the properties from the connected library are added to the list of properties of new documents or to the list of properties used to customize the report style in the Report style parameters dialog.
Add	The button allows to add a library to the list of libraries. It calls the standard Windows file opening dialog.
Delete	The button allows to delete a library from the list of libraries.

Tab. 9.2.9. Control elements of the Properties list setup dialog

	Element	Setting description
	Move Down/Up	This button allows to move a library selected in the list of libraries up or down by one position.
	Properties	This page allows to customize displaying of properties in the current document, new documents or the list of properties of the Report style parameters dialog.
	Properties List	<p>The list contains the system and advanced properties. A set of system properties is the same for all documents. A set of advanced properties is generated in the following way:</p> <ul style="list-style-type: none"> ▼ For the current document - the properties set by the user for this document are included into the set ▼ For new documents and reports— the properties from connected property libraries. <p>The following data are displayed for each property in the list of properties: name, information on the property source, and a comment specified for the property.</p> <p>The option to the left from the property name allows to control the displaying of this property in the current document, new documents or the list of properties of the Report style parameters dialog.</p>
	All	<p>This option allows to enable/disable the displaying of all properties contained in the list of properties. If the list contains both the displayed and non-displayed properties (not all options in the list are enabled), the All option will be displayed on the grey background. Clicking the option in such a status will disable it. Next — will enable it.</p>
	Preview	The button allows to view the parameters of a property selected in the list of properties. It calls the Property Parameters dialog.

Upon finishing setting up the list of properties, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.




9.2.6.6. Lines

9.2.6.6.1. Styles

Line styles use in graphic documents (both new and existing) is set up in dialogue **Line styles** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Lines — Styles**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Lines — Styles**.

Tab. 9.2.10. Control elements of the lines style list customization dialog

Element	Setting description
Line Filter	The group of control elements to generate a set of line styles available when creating and editing geometric objects.
Line styles list	This list contains line styles names and patterns. Enable options corresponding to styles which shall be included into the Style list on the Property bar when working with objects.
Enable All	This option allows enabling/disabling the availability of all layouts contained in the list. If the list contains both the available and non-available properties (not all options in the list are enabled), the All option will be displayed on the gray background. Clicking the option in such a status will disable it. Next — will enable it.
 Move Down/Up	The button allows to move the style selected in the list one position up or down.
	
 List re-arrangement	This button allows grouping styles activated for display in the upper part of the list.
Change List	This button allows adding and excluding user line styles to/from the list. After clicking this button the Line Style Manager is displayed (see Section 3.2.1.2.1 on p. 895).
Remember Last Style	This option controls the method of defining the default line style. <ul style="list-style-type: none"> ▼ If this option is activated, a line style from previous object is used for plotting each next geometrical object. ▼ If the option is disabled, then the line style that comes first in the list becomes the default one (“<i>By Default</i>” words are added to its name). If the very first or several styles coming first in the list are disabled, then the default one will be the first enabled style. The default style will be suggested each time after calling the command of creation of a geometric primitive. However, when you construct several primitives using the same command, the default drawing style for every next object is the line style used for the previous object.

Upon finishing line style settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.6.2. Center line

The default appearance of centerlines created by the commands **Centerline by Two Points**, **Auto Centerline** and **Center mark** is set up in **Centerline** dialog. The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Lines — Axis line,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Lines — Axis line.**

For enhanced convenience, the dialog includes an explanatory picture.

Tab. 9.2.11. Control elements of the system lines setup dialog.

Element	Setting description	Subgroup number*
1 – line outcome beyond the outline, mm	These fields allow entering necessary parameter values manually or setting them using counters.	2.2
2 – dotted line, mm		
3 – interval, mm		
4 – dash, mm	According to GOST, center lines must start and end with dashes. To fulfill this requirement, the dash lengths are proportionately changed when drawing lines. If the condition of having the dash length within the range prescribed by the standard (5-30 millimeters) shall be met when calculating it, select the auto determination option. If you need the dash length not exceeding a certain value (regardless of the standard requirements), select the max. length, mm option, and then type or set by the increment button the maximum length value.	2.2
5 – “cross” size, mm	The input field of the length of dashes making up a cross at the center of the object when creating a “cross”-shape center mark. In all other cases, the length of the central dashes will be determined by the system automatically.	2.2

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.



“Cross” size setting is also applied to a radial dimension with a kink.

Upon finishing with centerline display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.7. Split Line

Default parameters of break lines used graphic documents (both new and existing) are set up in dialogue **Break lines parameters** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Break line**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — Break Line**.

Tab. 9.2.12. Control elements of break line parameters setting dialog

Element	Setting description	Subgroup number**
Break Line Type	This group of options allows selecting a type of break line that will be used as the default to create view breaks.	2.1
Wave line height as % of length Amplitude, max	Fields for setting the height of wavy or bent line. See Section 3.5.5.8.3 on p. 1180 for detailed description of these fields.	2.1
Gap	This field serves to enter or select the distance between the break boundaries — the distance, at which the shown portions of the image will be after creating a break. The minimum gap value is 1mm. *	2.1

* When using bent break lines, the gap should be not less than the doubled setback amount. Therefore, if the default is less than the doubled maximum height (setback), then, when creating a break bounded by bent break lines, the default gap value is automatically increased.

** Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

When finished with setting up break line parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

Settings made in this dialog are stored in the document and will not be affected by its transfer to another workplace.

9.2.6.8. Break lines

9.2.6.8.1. Wave line

Wave line drawing in graphic documents (both new and existing) is set up in dialog **Wave line** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Lines — Wave line**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Lines — Wave line**.

This dialogue allows selecting the default wave amplitude and number of half-waves of a wavy line. For enhanced convenience, the dialog includes an explanatory picture.

Tab. 9.2.13. Control elements of the wavy lines setup dialog.

Element	Setting description	Subgroup number*
1-amplitude	Type or set with the spin button the wave amplitude in the Value field using the increment button. Specify amplitude units by selecting one of the variants: ▼ in % — the number typed in the Value field, shows the relation between the amplitude and wave length; ▼ in mm — the number typed in the Value field, shows wave amplitude in millimeters; Amplitude is a the distance between a mid-line and its most wave point.	2.2
Number of half-waves	Type or set with the increment button an integral number of half-waves.	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.8.2. Broken line

Broken line drawing in graphic documents (both new and existing) is set up in dialogue **Broken line** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Lines — Broken line**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Lines — Broken line**.

Allows you to select default amplitude, number and type of bends and to specify line jut beyond the start and end points. For enhanced convenience, the dialog includes an explanatory picture.

Tab. 9.2.14. Control elements to manage lines with bends

Element	Setting description	Subgroup number*
1-amplitude, mm	Type or set with the increment button the bend amplitude. The Amplitude is a distance between a line and its most distant bend point.	2.2

Tab. 9.2.14. Control elements to manage lines with bends

Element	Setting description	Subgroup number*
2-Line Jut beyond End Points, mm	Type or set with the increment button the line jut beyond the start and end points.	2.2
Number of bends	Type or set with the increment button an integral number of bends.	2.1
Type of bends	Select the version to display a bend: ▼ Type 1 — antisymmetric with two apexes, ▼ Type 2 — axisymmetric with three apexes,	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.



If the specified number of bends with the current parameters doesn't fit in between the line's end points, the maximum possible number of bends is created.

Upon completion of setting up parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.8.3. Line styles filter

Line styles filter setup for bluff lines in graphic documents (both new and existing) is performed in **Object lines style filter** dialogue. The dialog is invoked by the following command:



- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Lines — Line style filter**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Lines — Line style filter**.

The similar dialogue is used for line styles filter setup:

- ▼ marks/reference designations,
- ▼ braces.

Serves to specify which line styles will be available when creating objects, to define their order in the list and select the default.

Tab. 9.2.15. Control elements of the lines style filter customization dialog

Element	Setting description	Subgroup number*
List of Lines	<p>This list contains system line styles names and patterns. Those styles whose options next to their names are enabled are available in the document: those are put in the list from which you select the line style when creating an object.</p> <p>If you use a set of lines other than the default, change the settings by enabling or disabling options by the line names.</p> <p>The first line style in the list has no option. It can not be disabled, i.e. it is considered to be always enabled. Therefore it is always available during object creation.</p>	1.2
 	<p>Move Down/Up</p> <p>These buttons serve to set up the order of styles in the list.</p> <p>To alter the position of a style, select it and press the appropriate button. The selected style will move one position to the specified direction.</p> <p>The order of the lines sequence defined in this dialog is propagated to the list in which the line style is selected when creating an object.</p> <p>To speed up object creation in the document, we recommend assigning the first position to the line style, which is used most often, and after it position the rest of the used line styles in the decreasing frequency of their use.</p>	1.2
Enable All	<p>This option serves to enable or disable all line styles at once. If not all list options are enabled, then the option is displayed in the gray background. Clicking the option in such a status will disable it. Next — will enable it.</p> <p>The Basic line style is enabled regardless of state of the Enable All option.</p>	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and sub-groups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.9. Multiline

Multiline default parameters for graphic documents (both new and existing) are set up in dialogue **Multiline** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Multiline**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Multiline**.

Tab. 9.2.16. Control elements of multiline parameters setup dialog

Element	Setting description	Subgroup number*
Lines	Group of elements allows specifying the default set of the multiline lines. To add a line into the set click the Add Line button, to change a line click the Change Line button, to delete a line click the Delete Line button. After pressing the Add Line or Change Line buttons the dialog of the line setup will be displayed on the screen. Define the required line style and offset in relation to the base line.	2.1
Limiters 1 Limiters 2	The group of elements which defines the default view of a limiter of the multiline end segment and its parameters— height and drawing line style. Detailed description of limiters — See. Section 3.2.12.8 on p. 951.	2.1
Load from Template	The button for loading a set of lines and types of limiters of the multiline end segments from a template. For detailed information on a template — See Section 3.2.12.5 on p. 945. After pressing the Load from Template button the standard file opening dialog shall be displayed. Select a file name and click the Open button. In the Lines group, a set of lines shall be displayed; in Limiters 1 and Limiters 2 groups, types of end segment limiters and their settings shall be displayed. If it is necessary, you can change the settings received from the template.	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Having finished the multiline setup, press **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

Settings made in this dialog are stored in the document and will not be affected by its transfer to another workplace.

9.2.6.10. Dimensions

9.2.6.10.1. General settings

General settings of dimensions for graphic documents (both new and existing) are set up in dialogue **General settings** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — General settings**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — General settings**.

The similar dialogue is used for making general settings of patterns dimensions.

This dialogue serves to set up the default gap/length of extension lines for linear dimensions and the default format of displaying angular dimensions.

Tab. 9.2.17. Control elements of the common dimensions settings dialogue

Element	Setting description	Subgroup number*
Fix for Linear Dimensions	<p>When creating dimensions, the extension line may start at some distance from the snapping point. You can fix both the length of extension lines and the gap between the snap point and the start of the extension line. To do this, enter the desired value in the Gap or length, mm field.</p> <p>Only integral values of length (gap) are allowed. If you need to set the gap between the extension line and the dimension snap point by a specified value, activate the Gap of Extension Lines option.</p> <p>If, on the contrary, the entered value should define the length of the extension line, activate the Length of Extension Lines option.</p> <p>If the value in the Gap or Length, mm field is equal to zero, then the extension lines begin at the dimension snap points and can have an arbitrary length.</p>	2.1

Tab. 9.2.17. Control elements of the common dimensions settings dialogue

Element	Setting description	Subgroup number*
Display Format of Angular Dimensions	<p>Select the format for displaying the angular value in the dimension text of angular dimensions.</p> <p>If you need to have the angular values displayed in the format XX°XX'XX", enable the Degrees, Minutes, Seconds option.</p> <p>If you need to have the angular values displayed in the format XX.XXX°, enable the Decimal System option.</p> <p>Dimension value precision is defined by the parameters specified in the dimension text precision settings dialog (see Section 9.2.6.10.12 on p. 1781).</p>	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon finishing with general dimension settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.2. Parameters

Settings of dimensions drawing for graphic documents (both new and existing) are set up in dialogue **Dimensions drawing settings** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — Parameters**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Parameters**.

The similar dialogue is used for making patterns dimensions drawing settings.

Serves to set up dimension geometrical parameters: the extension lines projection beyond the dimension line, the arrow length and angle, the distance from the leader shelf to the text, etc. The customized parameters belong to subgroup 1.1 (see Section 9.2.1.1 on p. 1747).

For enhanced convenience, the dialog includes an explanatory picture.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of a KOMPAS-3D working session.

This setting affects the following document objects that have arrows or obliques in them:

- ▼ - Sizes of all types
- ▼ - Geometric tolerance symbols
- ▼ Surface finish symbols on shelf.

Upon finishing with dimension display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.3. Level marks

Settings of level marks drawing during height dimensions plotting for graphic documents (both new and existing) are set up in dialogue **Level marks drawing settings** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — Level marks,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Level marks.**

For enhanced convenience, the dialog includes an explanatory picture.

Tab. 9.2.18. Control elements of the level marks drawing setup dialog.

Element	Setting description	Subgroup number*
Title	This option allows selecting level marks' arrows drawing option. By default the option is disabled, and arrows are drawn in thin lines. To have the obliques displayed in normal lines, enable the Normal option. The change in the option state is reflected on the arrow example.	1.1
1 - arrow length, mm	This field allows entering or setting using a counter arrow length in mm.	1.1
2 - distance to shelf, mm	This field allows entering or setting using a counter a distance between an arrow and a shelf in mm.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon completion of arrow display setup, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.4. Arrows and obliques

The draw settings of arrows and obliques of dimensions for both new and existing graphic documents are set in **Arrows and obliques** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — Arrows and obliques,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Arrows and obliques.**

The similar dialogue is used in setting up dimensional arrows and ticks drawing for patterns.

Tab. 9.2.19. Dialog for Setting up Display of Arrows and ticks

Element	Setting description	Subgroup number*
Placing arrows	To select a dimension line arrow placement option relative to the extension lines, activate the desired option: Inside , Outside or Auto . The Auto option means the system will be automatically determine for each dimension whether to place the arrows inside or outside.	2.1
Dimension Line Arrows	To have the dimension arrows blackened, activate the Fill Arrows option. If this option is disabled, then arrows will be displayed in lines. The change in the option state is reflected on the arrow example. The Fill setting affects the following objects: <ul style="list-style-type: none"> ▼ All dimensions with the arrows of the following types: “Arrow”, “Solid Arrow”, “Point”, “Point-Small”; ▼ Geometric Tolerance symbols; ▼ Viewing direction arrows, ▼ Surface finish symbols on shelf. 	1.1
Obliques	To have the obliques displayed in normal lines, enable the Normal option. If this option is disabled, then obliques will be displayed in thin lines. Changes in the option status are reflected on the oblique example. The oblique drawing setting affects all objects that have (or may have) obliques as their elements: <ul style="list-style-type: none"> ▼ Dimensions with the oblique arrow type; ▼ Surface finish symbols on shelf. 	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon completion of arrows and obliques setup, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.5. Arrows filters — Linear dimensions

Settings of linear dimensional arrows drawing for graphic documents (both new and existing) are set up in dialogue **Dimensional arrows filter settings** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — Arrow filters — Linear dimensions**,



- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Dimensions — Arrow filters — Linear dimensions**,
Similar dialog is used for setting up the arrow filter:
 - ▼ in graphic documents,
 - ▼ circles and arches,
 - ▼ angular dimensions,
 - ▼ leaders,
 - ▼ reference number symbols,
 - ▼ marks/reference designations,
 - ▼ leader texts,
 - ▼ cut lines,
 - ▼ cut/section lines,
 - ▼ view arrows,
 - ▼ in models,
 - ▼ linear dimensions,
 - ▼ circles and arches dimensions,
 - ▼ angular dimensions,
 - ▼ leaders,
 - ▼ reference number symbols.

This dialogue allows specifying which arrows will be available when creating objects, define their order in the list and select the default.

Tab. 9.2.20. Control elements of the arrows filter setup dialog

Element	Setting description	Subgroup number*
Arrows List	<p>The list includes all arrow descriptions and patterns available in the arrows types system for the selected object type (dimension, leader or other designation). Those arrows which options next to their names are enabled are available in the document: they get in the list, from which the arrow type is selected at the time of creating an object.</p> <p>If you use a set of arrows other than the default, change the settings by enabling or disabling options by the arrow names.</p> <p>One arrow type— No Arrow— does not have an option. It can not be disabled, i.e. it is considered to be always enabled. Therefore, it is constantly available when creating objects in documents.</p>	1.2

Tab. 9.2.20. Control elements of the arrows filter setup dialog

Element	Setting description	Subgroup number*
 	<p>Move Down/Up</p> <p>These buttons serve to set up the order of arrows in the list.</p> <p>To alter the position of an arrow, select it and click the appropriate button. The selected arrow will move by one position in the specified direction.</p> <p>The order of the arrows sequence defined in this dialog is propagated to the list in which the arrow type is selected when creating an object. The arrow that comes first in the list becomes the default.</p> <p>If the very first arrow or several arrows coming first in the list are disabled, then the first enabled arrow becomes the default one.</p> <p>To speed up object creation in the document, we recommend assigning the first position to the arrow type, which is used most often, and after it position the rest of the used arrow types in the decreasing frequency of their use.</p>	1.2
	<p>Enable All</p> <p>This option serves to enable or disable all arrow types at once. If not all arrows are enabled, then the option is displayed in the gray background grayed out. Clicking the option in such a status will disable it. Next — will enable it.</p> <p>The No Arrow type of the arrow remains enabled regardless of the Enable All option state.</p>	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon finishing with arrow filter settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.6. Arrows filters— Circles and arcs dimensions

Settings of circles and arcs dimensional arrows drawing for graphic documents (both new and existing) are set up in dialogue **Circles and arcs dimensional arrows filter settings** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — Arrow filters — Dimensions of circles and arcs,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Dimensions — Arrow filters — Dimensions of circles and arcs,**

Work with dialog is described in Section 9.2.6.10.5.

9.2.6.10.7. Arrows filters – Angular dimensions

Settings of angular dimensional arrows drawing for graphic documents (both new and existing) are set up in dialogue **Angular dimensional arrows filter settings** called by the command:

- ▼ for new documents — **Service – Parameters... – New Documents – Graphic Document – Dimensions – Arrow filters – Angle dimensions,**
- ▼ for the current document — **Service – Parameters... – Current drawing/fragment – Dimensions – Dimensions – Arrow filters – Angle dimensions.**

Work with dialog is described in Section 9.2.6.10.5.

9.2.6.10.8. Inscription

Settings of dimensions inscriptions text parameters for graphic documents (both new and existing) are set up in dialogue **Dimensions inscription settings** called by the command:

- ▼ for new documents — **Service – Parameters... – New Documents – Graphic Document – Dimensions – Text,**
- ▼ for the current document — **Service – Parameters... – Current drawing/fragment – Dimensions – Text.**

Similar dialog is used for setting up text parameters:

- ▼ in graphic documents,
 - ▼ leaders texts,
 - ▼ reference number symbols,
 - ▼ modification symbols,
 - ▼ marks/reference designations,
 - ▼ node point numbers,
 - ▼ leader texts,
 - ▼ braces,
 - ▼ marks,
 - ▼ symbol of unspecified surface finish,
- ▼ in models,
 - ▼ dimensional texts,
 - ▼ leaders texts,
 - ▼ reference number symbols.

Tab. 9.2.21. Control elements of the inscription text options setup dialog

Element	Setting description	Subgroup number*
Font	The drop-down list allows you to select the desired font. The list contains all fonts enabled in Windows.	2.2

Tab. 9.2.21. Control elements of the inscription text options setup dialog

Element	Setting description	Subgroup number*
Different from Default Font	This option allows defining font correspondence or difference from default document font. If this option is disabled, font always matches the default one.	2.2
Height, mm	In the field, enter the characters height (in millimeters) or select this value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Compression	In the field, enter the symbols' compression value or select the value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Line spacing, mm	In the field, enter the text line spacing value. For some annotations, the line spacing doesn't matter; therefore this field is missing when setting them up.	2.2
Color...	The button allows to choose symbol color in the standard color selection dialog.	2.2
Italic	The option allows to set the italic style of characters.	2.2
Bold	The option allows to set the bold style of characters.	2.2
Underlined	The option allows to set the underlined style of characters.	2.2
Example	The preview window shows the text appearance. This helps to make visual evaluation of introduced changes.	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.9. Inscription position

Settings of dimensions inscriptions position for graphic documents (both new and existing) are set up in dialogue **Dimensions inscription position** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — Text position,**

- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Text position.**

A similar dialogue is used for dimensional texts position setup for models.

Serves to define the default position of the dimension text in linear and angular dimensions. Adjustable parameters belong to subgroup 2.1 (see Section 9.2.1.1 on p. 1747).

To select dimensional texts position option for linear dimensions, enable the necessary option in **Linear dimensions** group, for angular dimensions— in **Angular dimensions** group. The schemes of dimension text positions corresponding to the options are shown on the figures.

The selected version of the position for the linear dimension also determines the position of the diameter and radius dimensions.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.10. Tolerances and tolerance range — Parameters

Display parameters for tolerances and tolerance range of dimensions in graphic documents (both new and current) are set up in the **Parameters of tolerances and tolerance ranges** dialog called with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Dimensions — Tolerances and tolerance range — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Tolerances and tolerance range — Parameters.**

The similar dialogue is used in setting up Tolerances and limit values adjustment for patterns.

Tab. 9.2.22. Tolerances and limit values adjustment dialogue control elements

Element	Setting description	Subgroup number*
Classes of Accuracy	<p>This is the group of interface elements serving to manage the availability of main and auxiliary classes of accuracy when creating dimensions.</p> <p>If both options are selected, Primary and Auxiliary, then, when creating dimensions, the Primary and Auxiliary classes of accuracy will be available along with the preferred ones.</p> <p>If none of the options is selected, then you will be able to choose only from the preferred classes of accuracy.</p> <p>Whether accuracy class can be displayed in the caption is defined by a setting made in dimensions accuracy setting dialog (see Section 9.2.6.10.12). In it you can specify the number starting from which the class of accuracy is not shown in the dimension text.</p>	1.2

Tab. 9.2.22. Tolerances and limit values adjustment dialogue control elements

Element	Setting description	Subgroup number*
Include into Text	This group of items serves to control the default inclusion in the dimension text of classes of accuracies and the values defining the tolerance field: tolerances or values range.	2.1
Display Type	This is the group of interface elements serving to select the type of values defining the tolerance field. If maximum dimension deviations should be included in the dimension text, enable the option Deviations , and if you need the dimension limit values, then use the Limit values option. When the Limit values option is enabled, the Limit values in One Line option becomes available. When selecting it, the maximum and minimum dimension values are written not one on top of the other, but one next to the other, hyphen-separated.	2.1
Text Alignment	The group of options that serves to select the method of positioning deviations or limit values relative to the nominal value. Placement opportunities are shown on the figures. To select the default placement of values, enable the desired option.	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After finishing with the settings of tolerances and value ranges limits, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.11. Tolerances and tolerance range – Default tolerances

Default parameters of tolerances in graphic documents (both new and current ones) are set up in the **Value and state of default tolerances** dialog called by this command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Dimensions — Tolerances and tolerance range — Default tolerances**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Tolerances and tolerance range — Default tolerances**.

The dialog contains a group of buttons for **Default tolerances of dimensions**. The buttons are used to call the **Tolerance** dialog where the default value of tolerance is set either by specifying the class of accuracy or entering maximum deviations. Setup is performed separately for each type of dimensions: linear, angular, diameter, radial (only deviations are set up for angular

dimensions). Setup parameters belong to subgroup 2.1 (see Section 9.2.1.1 on p. 1747).

There is a field next to each button for previewing the current value of tolerance. The field cannot be edited manually; content variants:

- ▼ 10 with a class of accuracy or deviations¹ — dimensions of this type will have the specified tolerance,
- ▼ an empty field — dimensions of this type will have no tolerance.



The field can only contain 10. This means the tolerance is set by a class of accuracy but the number of the class of accuracy for which display in the dimension text is disabled has been selected. The accuracy class limiting number is specified during the setup of precision of dimension texts (see Section 9.2.6.10.12 on p. 1781).

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.10.12. Precision

Settings of dimensions values display parameters in dimension texts for graphic documents (both new and existing) are set up in dialogue **Dimensions inscription accuracy** called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Dimensions — Precision**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Dimensions — Precision**.

The similar dialogue is used in setting up dimension values display parameters adjustment for patterns.

Tab. 9.2.23. Control elements of dimension texts parameters setting dialog

Element	Setting description	Subgroup number*
Linear Dimensions	<p>A group of elements allowing to customize the display of decimal dimensions, i.e. linear, in xx,xxx format, and angular, in xx,xxx° format.</p> <p>Enter or set using a counter a number of decimal figures that shall be displayed in dimension texts in Number of decimal digits in dimensional texts field.</p> <p>Activate the Show Trailing Decimal Zeroes option to make the dimension text always display the specified number of decimal digits. In this case, padding zeros will be automatically added to dimension text.</p>	1.1

1. 10 is for illustration purposes.

Tab. 9.2.23. Control elements of dimension texts parameters setting dialog

Element	Setting description	Subgroup number*
Angular Dimensions	This group of options sets the displaying of angular units in $xx^\circ xx' xx''$ format. The angle values can be displayed with the accuracy up to degrees, minutes and seconds. To make the desired selection, enable the respective option: Degrees, Minutes, Seconds . Angular dimensions display format is set in the general dimensions settings dialogue (see Section 9.2.6.10.1 on p. 1771).	1.1
The Number, Starting from which the Class of Accuracy is not Added to the Dimension Text	This field allows quality assurance number entering or setting using a counter. For example, by entering the number <i>9</i> in this field, the names of all classes of accuracy with the numbers <i>9, 10, 11</i> , etc., will not be displayed in dimension texts. This interface item is accessible when the Show All option is disabled. If the Show All option is enabled, then you can include a class of accuracy with an arbitrary number into the dimension text.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.11. Leader

9.2.6.11.1. Parameters

Leader display set up for graphical documents (new and current) is carried out in **leader line display setting** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Leader line — Parameters**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Leader line — Parameters**.

The similar dialog is used when setting up header display parameters for models.

The Dialog makes it possible to set up leader geometrical parameters— the arrow length and angle, the distance from the leader shelf to the text, etc. Customizable parameters belong to subgroup 1.1 (see. Section 9.2.1.1 on p. 1747).

For enhanced convenience, the dialog includes an explanatory picture.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.

Upon finishing with leader line display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.11.2. Arrows and obliques

The display settings of arrows and obliques for both new and existing graphic documents are set in **Arrows and Obliques** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Leader line — Arrows and obliques**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Leader line — Arrows and obliques** command.

The similar dialog is used when setting up model arrows and obliques of marks/reference designations with a leader for graphic documents and leaders in models.

Tab. 9.2.24. Control elements of arrows and obliques setup dialog

Element	Setting description	Subgroup number*
Leader Arrows	To have the leader arrows blackened, activate the Fill Arrows option. If this option is disabled, then arrows will be displayed in lines. Changes in the option status are reflected on the arrow example in the dialog.	1.1
Leader Obliques	To have the obliques displayed in normal lines, enable the Normal option. If this option is disabled, then obliques will be displayed in thin lines. Changes in the option status are reflected on the oblique example.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Tab. 9.2.25. Control elements of the arrows and obliques setup dialog

Element	Setting description
Leader Arrows	To have the leader arrows blackened, activate the Fill Arrows option. If this option is disabled, then arrows will be displayed in lines. Changes in the option status are reflected on the arrow example in the dialog.
Leader Obliques	To have the obliques displayed in normal lines, enable the Normal option. If this option is disabled, then obliques will be displayed in thin lines. Changes in the option status are reflected on the oblique example.

Upon completion of arrows and obliques setup, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.11.3. Arrow filter

Leader display set up for graphical documents (new and current) is carried out in **leader line filter** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Leader line — Arrow filter**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Leader line — Arrow filter**.

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.6.11.4. Text Above/Below/After Leader Shelf

Leader text display set up for graphical documents (new and current) is carried out in **Text Above/Below/After Leader Shelf** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Leader line — Text above/under/behind the shelf**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Leader line — Text above/under/behind the shelf**,

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.11.5. Inclined Text

Inclined text display set up on leaders of graphical documents (new and current) is carried out in **Text Above/Below/After Leader Shelf** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Leader line — Inclined text**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Leader line — Inclined text**.

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.11.6. Symbol sizes

Setup of symbol sizes for graphical documents (new and current) is carried out in **Symbol sizes** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Leader line — Symbol dimensions**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Leader line — Symbol dimensions**.

The similar dialog is used for setting of Symbol sizes for models.

The dialog allows you to dimension circle of the marking symbol, the triangular stamping symbol and the size of symbols for bindings symbols for joining.

Tab. 9.2.26. Control elements of the symbol sizes setup dialog.

Element	Setting description	Subgroup number*
Circle diameter of marking symbol, mm	In the field, enter the circle diameter (in millimeters) of the marking symbol circles in a document. According to GOST 2.314-68 standard, the diameter must be from 10 to 15 millimeters.	1.1
Height of triangular stamping symbol, mm	In the field, enter the triangle height (in millimeters) of the marking symbol circles in a document. According to GOST 2.314-68, the height must be from 10 to 15 millimeters.	1.1
Size of symbols for Joining by soldering, bonding, sewing and metal stapling, in millimeters	In the input field, enter the size of symbols for bindings by Soldering, Bonding, Sewing and Metal Stapling (in millimeters) in a graphic document.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.



If the size of marking identifier and stamping signs will be too small, the text of those designations may run beyond the sign extents. To change the font in designations, use dialog of setting a text above/under leader shelf (see Section 9.2.6.11.4 on p. 1784).

Upon finishing symbol size settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.12. Conditional Crossing

Set up of the parameters of an intersection notation for graphical documents (new and current) is carried out in **Intersection notation** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Conditional crossing,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Conditional crossing.**

Tab. 9.2.27. Control elements of intersection notation parameters setup dialog

Element	Setting description	Subgroup number*
Extension of dimension lines beyond intersection point, mm	Use input field to set the value of origin for the extension lines (in millimeters) in the document.	1.1
To draw intersection point with style	The group of elements makes it possible to select way of display of extension lines' intersection point. If you want to set point definition in intersection notation, activate Draw intersection point with style option. Select style of drawing a point from the drop-down list.	2.1
Fix for intersection notation	Group of elements make it possible to fix gap or length of extension lines. The extension line may start at some distance from the object point. You can fix both the length of extension lines and the gap between the object and the start of the extension line. To do this, enter the desired value in the Gap or length, mm field. Only integral values of length (gap) are allowed. If you need to set the gap between an object and extension line by a specified value, activate the Gap of Extension Lines option. If, on the contrary, the entered value should define the length of the extension line, activate the Length of Extension Lines option. If the value in the Gap or length, mm field is equal to zero or greater than the distance between the object and intersection point, extension lines are built from ending points of the objects to the intersection point.	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon completion of setting up parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13. Designations for mechanical engineering

9.2.6.13.1. Reference number — Parameters

Ref. No. Display Set up for graphical documents (new and current) is carried out in **Reference number symbol display parameters** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of position — Parameters**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Designation of position — Parameters**.

The similar dialog is used when setting up display for graphic documents in models.

The dialog makes it possible to set up geometrical parameters of position designations—the arrow length and angle, the distance from the leader shelf to the text, etc. Customizable parameters belong to subgroup 1.1 (see Section 9.2.1.1 on p. 1747).

For enhanced convenience, the dialog includes an explanatory picture.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.

Upon finishing with the position designation display settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.2. Reference number — Form Parameters

Reference number display set up for graphical documents (new and current) is carried out in **Reference number display parameters** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of position — Form parameters**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Designation of position — Form parameters**.

The similar dialog is used when setting up display for graphic documents in models.

Tab. 9.2.28. Control elements of the position reference number setup dialog.

Element	Setting description	Subgroup number*
Form Size	<p>You can define the size for position reference number symbols with the shape type “Circle”, “Hexagon”, “Circle with divider”. Set required dimension value (in mm) manually in Value field or select it in the list that drops down.</p> <p>In the case of the “Circle” form type, this value is defined by the diameter, while for the “Hexagon” form type—the inscribed circle diameter, and for the “Circle with divider” form type — the radius.</p> <p>If the form size is too small, then the position reference numbers could run beyond the symbol extents. To modify the size of the position reference numbers font, use the dialog to set up position designations text. (see Section 9.2.6.13.5 on p. 1789).</p>	1.1
Form Parameters	Group of elements serves to define parameters of position reference numbers.	—
Shape Type	Required position reference number type can be selected in the dropdown list.	2.1
Horizontal	<p>This option controls the placement of a position reference number symbol that contains several position numbers.</p> <p>If the option is enabled, then numbers are arranged in a column, if enabled— in a row.</p>	2.1
Make shelf	<p>This option controls the display of the position leader shelf.</p> <p>In the case of reference number positions with the “Plain text” form type the shelf appears as a common leader shelf, with the text positioned above it. For other shape types, the shelf is a line segment connecting the text with the leader line.</p> <p>Examples of reference designation forms types are shown on Fig. 3.3.63 on p. 1035.</p>	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.3. Reference number position symbol— Arrows

Parameters for displaying arrows of reference numbers for graphic documents (both new and current) are set up in the **Arrows** dialog called by the command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of position — Arrows**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Designation of position — Arrows**.

The same dialog is used to setup drawing of cut/section lines' arrows, view arrows, cut lines' arrows for construction, as well as arrows of position designations in models.

The dialog contains **Fill** option. By default, this option is disabled. In this case, arrows will be displayed in lines. To have the position reference symbol arrows blackened, activate this option. Changes in the option status are reflected on the arrow example in the dialog.

Adjustable parameters — **Fill Arrows**— is related to subgroup1.1 (see Section 9.2.1.1 on p. 1747).

Upon completion of arrow display setup, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.4. Designation of position— Arrow filter

Reference number display set up for graphical documents (new and current) is carried out in **Position reference symbol arrow display parameters** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of position — Arrow filter**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Designation of position — Arrow filter**.

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.6.13.5. Position reference number symbol — Position symbol text:

Position reference number display parameters set up for graphic documents (new and current) is carried out in **Reference Number Display parameters** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of position — Position designation text**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Designation of position — Position designation text**.

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.13.6. Surface finish

Surface finish reference number display parameters set up for graphical documents (new and current) is carried out in **Parameters of surface finish designations** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of position — Surface finish,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Designation of position — Surface finish.**

The similar dialog is used when setting up designation for surface finish designation parameters in models.

Tab. 9.2.29. Control elements of surface finish designation setup dialog.

Element	Setting description	Subgroup number*
As in GOST 2.309-73	This group of options serves to select the structure of surface finish symbols. When changing the setting for the current document, all surface finish symbols contained in it are rebuilt. Therefore, to edit a drawing that uses surface finish symbols per previous edition of GOST 2.309-73, you need to open that drawing, change the setting of surface finish symbols and save it.	1.1
Text Parameters	Group of elements allows to set up a text in surface finish designations — enter required line spacing (in mm) and set font parameters.	2.2

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.



Arrow filling and oblique line thickness (for on-shelf finish designation) are controlled by the dimension setting (see Section).



Text settings made in the current dialog affect all surface finish designations set on the images of surfaces. The setting of the unspecified surface finish text is done in the section **Document parameters— Unspecified surface finish** of the settings dialogs for the current drawing and new drawings. (see. Section 9.2.6.18.7 on p. 1827).

Upon finishing with surface finish display parameters settings, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

9.2.6.13.7. Shape Deviations and Datum

The parameters of shape deviation and datum drawing for graphic documents (both new and current) are set up in the **Parameters of shape deviations and datum** dialog called by the :

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Parameters of shape deviations and datum,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for mechanical engineering — Parameters of shape deviations and datum.**

The similar dialog is used when setting up shape and datum deviations display settings in models.

Serves to assign the parameters that will be used by default when applying geometric tolerances and datum symbols.

Tab. 9.2.30. Control elements of shape and datum deviations setup dialog

Element	Setting description	Subgroup number*
Fill Triangle	The option allows to fill triangle when drawing a datum or geometric tolerance symbol. Arrow filling is controlled by the dimension setting.	1.1
Name	The drop-down list allows you to select the desired font. The list contains all fonts enabled in Windows.	2.2
Different from Default Font	This option allows defining font correspondence or difference from default document font. If this option is disabled, font always matches the default one.	2.2
Height, mm	In the field, enter the characters height (in millimeters) or select this value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Compression	In the field, enter the symbols' compression value or select the value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Color...	The button allows to choose symbol color in the standard color selection dialog.	2.2
Italic	The option allows to set the italic style of characters.	2.2
Bold	The option allows to set the bold style of characters.	2.2

Tab. 9.2.30. Control elements of shape and datum deviations setup dialog

Element	Setting description	Subgroup number*
Underlined	The option allows to set the underlined style of characters.	2.2
Example	The preview window shows the text appearance. This helps to make visual evaluation of introduced changes.	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.



The height of the triangle is automatically defined by the system. According to GOST2.308-79, it is equal to the font height of dimension values.

The height of the frame is defined automatically, as well. In correspondence to Appendix 1 to GOST 2.308-79 it is equal to doubled height of the font used in the shape's tolerance text.

Upon completion of setting up parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.8. Cut/section line — Parameters

Setting of parameters of cutting /section line for graphical documents (new and current) is done in **Cut/Section Line settings** dialog, invoked by the command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Cut/section line — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Cut/section line — Parameters.**

The similar dialog is used when setting up parameters of the cut line for construction.

Tab. 9.2.31. Control elements of cut lines /section setup dialog

Element	Setting description	Subgroup number*
Name	The drop-down list allows you to select the desired font. The list contains all fonts enabled in Windows.	2.2
Different from Default Font	This option allows defining font correspondence or difference from default document font. If this option is disabled, font always matches the default one.	2.2

Tab. 9.2.31. Control elements of cut lines /section setup dialog

Element	Setting description	Subgroup number*
Height, mm	In the field, enter the characters height (in millimeters) or select this value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Compression	In the field, enter the symbols' compression value or select the value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Color...	The button allows to choose symbol color in the standard color selection dialog.	2.2
Italic	The option allows to set the italic style of characters.	2.2
Bold	The option allows to set the bold style of characters.	2.2
Underlined	The option allows to set the underlined style of characters.	2.2
Example	The preview window shows the text appearance. This helps to make visual evaluation of introduced changes.	—
Dash length, mm	Enter the length of a cut/section line dash (in millimeters).	1.1
Text for designating	This field serves for entering the text to be inserted before the cut number when creating links to the cut designation. The text changes made will be transferred to the already created links. This field is available in the dialog when setting the parameters of the cut line for construction.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.9. Cut/section line — Arrows

Parameters for displaying arrows of cut/section lines for graphic documents (both new and current) are set up in the **Arrows** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Cut/section line — Arrows,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Cut/section line — Arrows.**

Work with dialog is described in Section 9.2.6.13.3 on p. 1789.

9.2.6.13.10. Cut/section line — Arrow filter

Setting of arrow filter of cutting /section line for graphical documents (new and current) is done in **Cut/Section Line Arrow Filter** dialog, invoked by the command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Cut/section line — Arrow Filter,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Cut/section line — Arrow filter.**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.6.13.11. View arrow — Parameters

View arrow parameters set up for graphical documents (new and current) is carried out in **View arrow parameters** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Parameters.**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Parameters.**

Tab. 9.2.32. Control elements of view arrow setup dialog

Element	Setting description	Subgroup number*
Name	The drop-down list allows you to select the desired font. The list contains all fonts enabled in Windows.	2.2
Different from Default Font	This option allows defining font correspondence or difference from default document font. If this option is disabled, font always matches the default one.	2.2
Height, mm	In the field, enter the characters height (in millimeters) or select this value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2

Tab. 9.2.32. Control elements of view arrow setup dialog

Element	Setting description	Subgroup number*
Compression	In the field, enter the symbols' compression value or select the value from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Color...	The button allows to choose symbol color in the standard color selection dialog.	2.2
Italic	The option allows to set the italic style of characters.	2.2
Bold	The option allows to set the bold style of characters.	2.2
Underlined	The option allows to set the underlined style of characters.	2.2
Example	The preview window shows the text appearance. This helps to make visual evaluation of introduced changes.	—
Object length, mm	This is the field to enter the length of the segment which is part of the view direction arrow.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.12. View arrow — Arrows

Parameters for displaying arrows of view directions for graphic documents (both new and current) are set up in the **Arrows** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — View arrow — Arrows**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — View arrow — Arrows**. Work with dialog is described in Section 9.2.6.13.3 on p. 1789.

9.2.6.13.13. View arrow — Arrow filter

View arrow filter set up for graphical documents (new and current) is carried out in **View arrow filter** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — View arrow — Arrow filter**,

- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — View arrow — Arrow filter.**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.6.13.14. Auto -sorting

Enabling and setting up automatic sorting for graphical documents (new and current) is carried out in **Auto-sorting of lettered items** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Auto sorting,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Auto sorting.**

The similar dialog is used when setting up auto-sorting in models.

Tab. 9.2.33. Control elements of auto-sorting setup dialog

Element	Setting description	Subgroup number**
Sorting Letters	The list defining which letters and in which order will be assigned to the layout objects. This field allows only letter input (whether upper or lower case). Duplicate letter use is not allowed. The letters specified in this field are displayed in symbol selection menu (see. Fig. 3.3.73 on p. 1044).	1.1
Auto-Sorting	The option serves to toggle on or off the auto-sorting mode of letter designations. When the option is enabled the Sorting order layout objects group is available.	—
Sorting order of layout objects	This is a group of control elements to specify which objects and in which order will be automatically assigned with letter designations. The list displays the names of layout object types, whose designations can be automatically sorted*. To enable Auto-sorting of desired-type object designations, activate the option before its name. Auto-sorting can also be controlled by the Auto-Sorting option available on the Property Bar when creating objects of the respective types. The priority of objects of different types is determined by the position of the respective name in the list. To raise required object priority, select it in the list and click the Move up button, to lower it, click the Move down button. The selected name will move one position to the specified direction.	1.1

- * When a model document is being adjusted, the list contains a single object type: the datum symbol.
- ** Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon finishing the setup of auto-sorting, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.15. Modification symbol— Parameters

Modification symbol parameters set up for graphical documents (new and current) is carried out in **Modification symbol parameters** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of changes— Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Graphic document — Designations for mechanical engineering — Designation of change — Parameters.**

Tab. 9.2.34. Control elements of modification symbol setup dialog.

Element	Setting description	Subgroup number*
Symbol type	Group of options allows to choose modification symbol — square, circle, parentheses . For a parentheses symbol, select the type of parentheses from the drop-down list.	2.1
Size of modification symbols, mm	In the field, enter the size of the modification symbol: the box side length (for a square symbol) or circle diameter (for circle symbol). For parentheses the height of parentheses is defined by the size of the modification symbol text font. When selecting this symbol type Height of modification symbols, mm is not available.	2.1
Leader type	The group of options allows to select a method of setting lengths of modification symbol leader lines.	—
full length	Enable this option, in order to make arbitrary lengths of leader lines.	2.1
limited line, mm	Enable this option in order to make fixed length of all leader lines. When selecting this option, the length value input becomes accessible.	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.13.16. Modification symbol — Text

Modification symbol text parameters set up for graphical documents (new and current) is carried out in **Modification text parameters** dialog, invoked by command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for mechanical engineering — Designation of changes — Text**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for mechanical engineering — Designation of change — Text**.

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.14. Designations for civil engineering

9.2.6.14.1. Mark/positional designation— Common settings

Common settings of marks/positional designations for both new and existing graphic documents are performed in **Common settings** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — General settings**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Mark/positional designation — General settings**.

This dialog allows to create a separator— one or several characters — between the text of a mark and its number and also to enable smart numeration. Customizable parameters are included into following subgroups (see Section 9.2.1.1 on p. 1747):

- ▼ **Separator between Mark Text and its Number** — to subgroup 1.1,
- ▼ **Autonumbering** — to subgroup 1.2.

Enter characters into the **Separator between mark text and its number** field. Maximum number of symbols is three. These characters will be automatically pasted between mark text and its number. If one of designation element is absent (mark name or its number) then the separator is not displayed. The view port of the mark/positional designation text input (see Section 3.3.13.3.1 on p. 1071) dialog displays mark text along with the separator.



The separator is not part of the text and cannot be edited in the text input dialog.

Enable the **Smart Numeration** option to have the next sequential number automatically assigned to a mark upon creating of each new mark. Auto indexing is allowed for marks with same text designation. Mark type— **with a leader**, **on a line** or **without a leader**— is of no importance.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.2. Mark/positional designation— With Leader

Parameters

Display parameters of marks/positional designations with leader for both new and existing graphic documents are set in **Mark/Positional Designation display settings** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — With leader — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Mark/positional designation — With leader — Parameters.**

This dialog allows to set up geometrical parameters of leader lines—the arrow length and angle, the distance from the leader shelf to the text, etc. Customizable parameters are included into subgroup 1.1 (see Section 9.2.1.1 on p. 1747).

For enhanced convenience, the dialog includes explanatory pictures.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.

Upon completion of setting up parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

Arrows and obliques

The draw settings of arrows and obliques of leaders for graphic documents (for both new and current) are set in the **Arrows and obliques** dialog called by the command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — With leader — Arrows and obliques,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Mark/positional designation — With leader — Arrows and obliques.**

Working with the dialog is described in Section 9.2.6.11.2 on p. 1783.

Arrow filter

Settings of arrow filter for both new and existing graphic documents are performed in **Arrow filter for mark/positional designations** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — With leader — Arrow filter,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations — Designations for construction — Mark/positional designation — With leader — Arrow filter,**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

Text

Parameters of text on the leader of marks/positional designations for both new and existing graphic documents are set in **Mark/Positional Designation text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — With leader — Text,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations — Designations for construction — Mark/positional designation — With leader — Text,**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.14.3. Mark/positional designation — On line

Parameters

Parameters of marks/positional designations on line for both new and existing graphic documents are set in **Mark/Positional Designation parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — On line — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Mark/positional designation — On line — Parameters.**

The dialog allows to specify distances from the text to a line below or above it. Adjustable parameters belong to subgroup 1.1 (see Section 9.2.1.1 on p. 1747).

For enhanced convenience, the dialog includes an explanatory picture.



If upon creation of mark/positional designation a two-lined text is entered and its **On a Line**, placement method is selected, then the distance between the lines will be equal to the sum of distances from an upper and a bottom line to the line.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.

Upon completion of setting up display parameters, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

Text

Parameters of text on line of marks/positional designations for both new and existing graphic documents are set in **Mark/Positional Designation text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — On line — Text,**

- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations — Designations for construction — Mark/positional designation — On line — Text,**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.14.4. Mark/positional designation — Without leader

Parameters

Parameters of marks/positional designations without leader for both new and existing graphic documents are set in **Mark/Positional Designation parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — Without leader — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations — Designations for construction — Mark/positional designation — Without leader — Parameters,**

The dialog allows to set the shape dimensions for designations of different types and distance from the text to a shape separator. For enhanced convenience, the dialog includes explanatory pictures.

Tab. 9.2.35. Control elements of Mark/Positional Designation parameters setup dialog

Element	Setting description	Subgroup number*
Shape Dimensions	<p>The group of the elements allows to set dimensions for various shapes of a mark/positional designation. For this, do the following.</p> <p>Select the Shape type from the drop-down list.</p> <p>Specify the dimensions of this shape.</p> <ul style="list-style-type: none"> ▼ If the selected shape demands defining of one overall dimension, its value is set or entered using the counter in the field 1 — the Overall Dimensions, mm. In the case of double circle the value set will define the diameter of internal circle. The gap between the external and the internal circles is 1 mm. ▼ If the selected shape demands defining of two overall dimensions, their values are set or entered using the counter in 1— Width, mm and 2– Height, mm fields. In case of a rounded rectangle choice in the field 1 — Width, mm — is entered distance center to center semicircles. <p>Note that only shape dimensions are set in this dialog. Default shape and list of shapes available during the creation of objects, are set in Mark/positional designations shape filter dialog (see Section Form filter on page 2043).</p>	2.2
Text Location	<p>The group of the elements allows to set distance from the text to a shape separator.</p> <p>The separator— horizontal line in the middle of the shape— is created automatically when a two-line text is entered into designation. The first text row is arranged above the separator and the second one— below.</p> <p>Enter the distances into Distance from shape separator to text above/below, mm fields or choose distances from drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.</p>	2.2

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

Line styles filter

Settings of line styles filter for marks/positional designations for both new and existing graphic documents are performed in **Line style filter** dialog. The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — Without leader — Line style filter,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations — Designations for construction — Mark/positional designation — Without leader — Line style filter,**

Working with the dialog is described in Section 9.2.6.8.3 on p. 1768.



Shape filter

Settings of shape filter for marks/positional designations without leader for both new and existing graphic documents are performed in **Shape filter for mark/positional designations** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — Without leader — Form filter,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations — Designations for construction — Mark/positional designation — Without leader — Form filter,**

This dialog allows to specify shapes that will be available when creating objects, define their order in the list and select the default choice.

Tab. 9.2.36. Control elements of the shape filter setup dialog

Element	Setting description	Subgroup number*
Shape Listing	<p>This list contains names and examples of all shapes for marks/positional designations available (shape parameter settings are described at Section Parameters on page 2041).</p> <p>Those shapes whose options next to their names are enabled are available in the document: they get in the list, from which the designation shape is selected at the time of creating an object.</p> <p>If you use a set of shapes other than the default, change the settings by enabling or disabling options next to the shape names.</p> <p>A type of designation display is included in the list of shapes—Without Shape— which has no option. It can not be disabled, i.e. it is considered to be always enabled. Therefore, it is constantly available when creating objects in graphic documents.</p>	1.2
 	<p>Move Down/Up</p> <p>These buttons allow to set up the order of shapes in the list.</p> <p>To alter the position of a shape, select it and press the appropriate button. The selected shape will move by one position in the specified direction.</p> <p>The order of the shape sequence defined in this dialog is transferred to the list in which the shape of mark/positional designation is selected when creating an object. The shape that comes first in the list becomes the default.</p> <p>To speed up object creation in the document, it is recommended to assign the first position to the shape type used most often, and to position the rest of the used shape types after it in the decreasing order of usage.</p>	1.2
Enable All	<p>This option allows to enable or disable all shapes at once. If not all list options are enabled, then the option is displayed in the gray background. Clicking the option in this status will disable it, the next click will enable it.</p> <p>The Without Shape is enabled regardless of state of the Enable All option.</p>	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon completion of setting up line filter, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

Text

Parameters of text of marks/positional designations without leader for both new and existing graphic documents are set in **Mark/Positional Designation text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Mark/positional designation — Without leader — Text,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations — Designations for construction — Mark/positional designation — Without leader — Text,**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.14.5. Cut line — Parameters

Parameters of cut lines for both new and existing graphic documents are set in **Cut line parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Cut line — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Cut line — Parameters.**

Work with dialog is described in Section 9.2.6.13.8 on p. 1792.

9.2.6.14.6. Cut line — Arrows

Parameters for displaying arrows of cut lines for graphic documents (both new and current) are set up in the **Arrows** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Cut line — Arrows,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Cut line — Arrows.**

Working with dialog is described in Section 9.2.6.13.3 on p. 1789.

9.2.6.14.7. Cut line — Arrow filter

Setting of arrow filter of cutting /section lines for graphical documents (new and current) is done in **Cut Line Arrow Filter** dialog, invoked by the command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Cut line — Arrow filter,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Cut line — Arrow filter.**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.6.14.8. Node and sectional node designation— Common settings

Common settings of sectional node designation for both new and existing graphic documents are performed in **Common settings** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Node and sectional node designation — General settings**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Node and sectional node designation — General settings**,

Allows to select styles of a dash line, and also to specify, whether it is required to connect dashes a line, and to select style of this line. For enhanced convenience, the dialog includes explanatory pictures for parameters.

Tab. 9.2.37. Control elements of the node designations setup dialog

Element	Setting description	Subgroup number*
Dash Line Style	Group of options allow to choose dash line style — Normal or Thick . Option switch is reflected on the explanatory picture.	1.1
Line between Dashes	Group of options allows to choose the style of line between dashes— Thin or Dash-dotted , as well as switch off line drawing by choosing Do not draw . Option switch is reflected on the explanatory picture. Please note that the line is not displayed if a distance between dashes is less than the doubled value of a gap set in the Node designations setup dialog (see Section 9.2.6.14.9).	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.9. Node and sectional node designation — Parameters

Display parameters of Node and sectional node designation for both new and existing graphic documents are set in **Node and sectional node designation setup** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Node and sectional node designation — Parameters**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Node and sectional node designation — Parameters**,

The dialog allows to set select dash and text placement parameters in the node designations. **Dash length** parameters belongs to subgroup 2.1, all other parameters — to subgroup 1.1 (see Section 9.2.1.1 on p. 1747).

For enhanced convenience, the dialog includes explanatory pictures.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.

Upon completion of setting up display parameters, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.10. Node and sectional node designation — Text

Text parameters of node and sectional node designation for both new and existing graphic documents are set in **Node designation text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Node and sectional node designation — Text**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Node and sectional node designation — Text**,

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.14.11. Node number — Parameters

Node number parameters for both new and existing graphic documents are set in **Node number display parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Node number — Parameters**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Node number — Parameters**.

Allows to choose the shape of node number, set shape dimensions and distance from the text to a shape separator. For enhanced convenience, the dialog includes an explanatory picture.

Tab. 9.2.38. Control elements of the node number parameters setup dialog

Element	Setting description	Subgroup number*
Shape Type	This drop-down list serves to choose default shape type for node number designation. Shape type defines a number of circles and line styles of the circles display.	1.1
1 - Shape Size, mm	This field allows to enter or select from the list the circle diameter for node number designation. In the case of double circle the selected value will define the diameter of internal circle.	1.1

Tab. 9.2.38. Control elements of the node number parameters setup dialog

Element	Setting description	Subgroup number*
2 - Ring Width, mm	This field allows to enter or select from the list the ring width— distance between node number designation circles.	1.1
3 - The Distance from the Separator to the Text above, mm	The field allows to enter or select the distance between the text and the shape separator from the list. A separator — horizontal line in the middle of the shape—will be generated automatically if the node number text is two-lined. Separator line style— Thin.	1.1
4 - The Distance from the Separator to the Text under, mm		

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

A manually entered values are added to the list and saved for the duration of KOMPAS-3D working session.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.12. Node number— Text

Node number text parameters for both new and existing graphic documents are set in **Node number text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Node number — Text,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Node number — Text.**

Tab. 9.2.39. Control elements of the node number text parameters setup dialog

Element	Setting description	Subgroup number*
Font	The drop-down list allows you to select the desired font. The list contains all fonts enabled in Windows.	2.2
Different from Default Font	This option allows defining font correspondence or difference from default document font. If this option is disabled, font always matches the default one.	2.2

Tab. 9.2.39. Control elements of the node number text parameters setup dialog

Element	Setting description	Subgroup number*
Height, mm	The field serves to enter or select list the height of text template symbols (in millimeters) for single or two-lined text from the drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Compression	A field that allows to enter or select compression of symbols from drop-down list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Color...	The button allows to choose symbol color in the standard color selection dialog.	2.2
Italic	The option allows to set the italic style of characters.	2.2
Bold	The option allows to set the bold style of characters.	2.2
Underlined	The option allows to set the underlined style of characters.	2.2
Example	The preview window shows the symbol appearance. This helps to make visual evaluation of introduced changes.	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon completion of setting up parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.13. Leader text — General settings

Common settings of leader texts for both new and existing graphic documents are performed in **Common settings** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Leader text — General settings,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Leader text — General settings,**

This dialog allows setting up the default parameters defining the shape and alignment of leader text flanges. For enhanced convenience, the dialog includes explanatory pictures.

Tab. 9.2.40. Control elements of the leader text editing setup dialog

Element	Setting description	Subgroup number*
Shape Type	The group of options allows to select the shape of leader text. When Type 1 is selected, the upper flange is displayed with an additional vertical segment. When Type 2 is selected, the upper flange is displayed with no additional segment.	2.1
Flanges alignment	The group of options allows to select a method of defining length of flanges. With the Equal Length is selected, the length of all flanges will be aligned by the longest one. With the By Text Length is selected, the length of each flange will be aligned by length of text on it. Jut of flange beyond the text is to be adjusted in the Set up Leader Text Display dialog (see Section 9.2.6.14.14).	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Press the **OK** button upon completion of general leader text setup. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.14. Leader text— Parameters

Leader text parameters for both new and existing graphic documents are set in **Leader text display parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Leader text — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Leader text — Parameters,**

This dialog allows selecting the parameters of obliques and arrows as well as parameters defining text arrangement on the leader flanges. Adjustable parameters belong to subgroup 1.1 (see Section 9.2.1.1 on p. 1747).

For enhanced convenience, the dialog includes explanatory pictures.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.15. Leader text— Arrows and obliques

The display settings of arrows and obliques of leader texts for graphic documents (for both new and current) are set in the **Arrows and obliques** dialog called by the command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Leader text — Arrows and obliques,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Leader text — Arrows and obliques.**

Tab. 9.2.41. Control elements of arrows and obliques setup dialog

Element	Setting description	Subgroup number*
Arrows of leader texts	To have the leader arrows blackened, activate the Blacken option. If this option is disabled, then arrows will be displayed in lines. Changes in the option status are reflected on the arrow example in the dialog.	1.1
Obliques of leader texts	To have the obliques displayed in normal lines, enable the Normal option. If this option is disabled, then obliques will be displayed in thin lines. Changes in the option status are reflected on the oblique example.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.16. Leader text— Arrow filter

Parameters of arrow filter of leader texts for both new and existing graphic documents are set in **Arrow filter for leader texts** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Leader text — Arrow filter,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Leader text — Arrow filter,**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.6.14.17. Leader text — Text

Leader text parameters for both new and existing graphic documents are set in **Leader text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Leader text — Text,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Leader text — Text.**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.14.18. Brace— Common settings

Common settings of braces for both new and existing graphic documents are performed in **Common settings** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Brace — General settings,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Brace — General settings.**

This dialog allows to define a default type of text placement and a brace direction.

Tab. 9.2.42. Control elements of brace editing setup dialog

Element	Setting description	Subgroup number*
Text Location	This group of options allows specifying the desired choice of text placement. If On a Flange is selected, the text will be located on a leader flange. If the Smart is selected, the text will be located outside of the brace— as an inscription parallel to the brace.	2.1
Brace Direction	The group of options allows to select a brace direction. If a Direction 1 is selected, the brace will be located to the right of the vector routed from the first brace snap point to the second one. If a Direction 2 is selected, then the brace will be located to the left of the vector routed from the first brace snap point to the second one.	2.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.19. Brace — Parameters

Brace display parameters for both new and existing graphic documents are set in **Brace display parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Brace — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Brace — Parameters.**

The dialog allows to set brace rounding radius and text arrangement parameters. **Brace rounding radius** parameter belongs to subgroup 2.2, text arrangement parameters— to subgroup 1.1 (see Section 9.2.1.1 on p. 1747).

Two choices of text arrangement are available.

- ▼ Outside of the brace, as an inscription with lines parallel to the brace. The text is automatically centered with respect to the brace.
- ▼ On the leader flange. The leader starts from the middle of the brace.

For enhanced convenience, the dialog includes explanatory pictures.

Enter parameter values into dialog fields or select them from the drop-down lists. Entered values will be added to the lists and saved for the duration of KOMPAS-3D working session.

Upon completion of setting up display parameters, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.20. Brace — Line style filter

Settings of line styles filter for braces for both new and existing graphic documents are performed in **Line style filter** dialog. The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Brace — Line style filter**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Brace — Filter line style**.

Working with the dialog is described in Section 9.2.6.8.3 on p. 1768.

9.2.6.14.21. Brace— Text

Brace text parameters for both new and existing graphic documents are set in **Brace text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Brace — Text**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Brace — Text**.

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.14.22. Coordinate Axes — Common Settings

Common settings of coordinate axes display for both new and existing graphic documents are performed in **Common settings** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Coordinate axes — General settings**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Coordinate axes — General settings**.

Allows to set a dimension of a mark and the list of characters for designations of axes, and also to select an end of an axis where a mark will be displayed.

Tab. 9.2.43. Control elements of coordinate axes editing setup dialog

Element	Setting description	Subgroup number*
Marks	The group of the elements allows to set dimensions and location of a mark.	—
1 - Shape Size, mm	This field allows to enter or select from the list the circle diameter for mark designation. In the case of double circle the selected value defines the diameter of selected external circle. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
2 - Ring Width, mm	This field allows to enter or select from the list the ring width— distance between mark designation circles. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	2.2
Enable Mark 1, Enable Mark 2	Enable the Enable Mark 1 option to have an axis displayed with a mark located on a jut beyond the start point (p1). Enable the Enable Mark 2 option to have an axis displayed with a mark located on a jut beyond the end point(p2). If both options are disabled, the axis will be displayed without marks. Enabling one of these options— Enable Mark 1 or Enable Mark 2 — is sufficient for circular axes to have an axis displayed with a mark.	1.1
Designation Characters	This field allows to enter the list of characters used for mark designation. The characters are listed in order of usage. After all characters from the list are assigned, the dual characters taken one after another from the list of single characters will automatically start to be used.	1.2

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.23. Coordinate Axes— Parameters

Coordinate axes display parameters for both new and existing graphic documents are set in **Coordinate axes display parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Coordinate axes — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Coordinate axes — Parameters.**

Allows to select default dimensions of a coordination axes elements and to set parameters of a dash-dotted line. For enhanced convenience, the dialog includes explanatory pictures.

Tab. 9.2.44. Control elements of the coordinate axes setup dialog

Element	Setting description	Subgroup number*
1 - axis jut from a start point, mm 2 - axis jut from an end point, mm 3-4 - bend offset, % 5 - dotted line 6 - space	The group of fields allows to enter manually or set the values of coordinate axes parameters using the counter.	2.2
7 - dash	<p>A group of elements allowing to choose the method to determine the stroke length during the display.</p> <p>Enable the Smart Determination of Length option for dash length to be defined automatically. If this option is enabled, a dash length is calculated automatically so that it will allow axis start and end points to contain a dash, rather than a dot or a space.</p> <p>To ensure that dash length does not exceed a specified value, enable Length not more than, mm option. Dash length input field will become available. Enter maximum dash length or set it by counter.</p>	2.2

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.24. Coordinate Axes— Orientation Marker

Display parameters of orientation markers for coordinate axes for both new and existing graphic documents are set in **Orientation marker display parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Coordinate axes — Orientation marker**,
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Coordinate axes — Orientation marker**.

The dialog allows to set up dimensions and appearance of coordination axis orientation pointer arrows. For enhanced convenience, the dialog includes explanatory pictures.

Tab. 9.2.45. Control elements of orientation marker setup dialog

Element	Setting description	Subgroup number [*]
1 - length, mm	The field that allows you to input or set (with the counter) the pointer length.	2.2
2 - arrow length, mm	This field allows to enter pointer arrow length or select it from the list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	1.1
3 - arrow angle, deg	This field allows to enter pointer arrow angle or select it from the list. Manually entered value is added to the list and saved for the duration of the current KOMPAS-3D working session.	1.1
4 - length, mm	The field that allows you to input or set (with the counter) the gap between pointer and mark designation.	1.1
Arrows	This group of elements allows to select pointer arrow type. To choose the method of display, enable the corresponding option — Arrow , Solid Arrow or Open Arrow . To use arrow blackening, enable the Blacken option. If this option is disabled, then arrows will be displayed in lines. Arrow blackening is available for the following arrow types: Arrow and Solid Arrow . The change in the option status is reflected on the arrow example in the Example field.	1.1

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

Upon completion of setting up orientation pointer display, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.14.25. Coordinate Axes— Text

Parameters of text of marks for coordinate axes for both new and existing graphic documents are set in **Mark text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Designations for construction — Coordinate axes — Text,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Designations for construction — Coordinate axes — Text.**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.6.15. Text on a drawing

Text paragraph offset parameters for both new and existing graphic documents are set in **Parameters of text on a drawing** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Drawing text,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Drawing text.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

9.2.6.16. Table parameters

9.2.6.16.1. Table header

Table header text parameters for both new and existing graphic documents are set in **Table header text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Table heading,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Table heading.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

9.2.6.16.2. Table cell

Table cell text parameters for both new and existing graphic documents are set in **Table cell text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Table cell,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Table cell.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

9.2.6.16.3. Table name

Parameters of the name of the table on a sheet for graphical documents (both new and current) can be set up in the **Table name text parameters** dialog called by the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Table parameters — Table name,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Table parameters — Table name.**

The dialog is used to set up default text parameters of the table name (including for a non-associative report) and the distance between the table and the name (see Section 4.2.2.1.3 on p. 1364 on table names).



Similar parameters for associative tables are set at report setup (see Section 9.2.6.23.3 on p. 1835).

Tab. 9.2.46. Control elements of the table setup options dialog

Element	Setting description
Line spacing, mm	The field allows to set the distance between text lines.
First line Indent, mm	This field allows to set the value of the indentation to the right from the start of a paragraph in its first line.
Font...	A button to call the font parameter setup dialog.
Tabs...	A button to call the dialog to set up tabulation parameters.
Indentations	This group of fields allow to set up values of left and right indentations of paragraph (in millimeters). Table name paragraph borders are vertical lines on which its right and left borders are positioned. These lines are used to mark off the left indents, right indents and first line indent. In other words, the maximum name paragraph width is equal to the table width.
Spacing	This group of fields allow to set up values of spacing before and after the paragraph (in millimeters).
Alignment	This group of fields allows to define the desired paragraph alignment method (left, right, centered, justified).
Distance to table, mm	This field allows to set up the required distance between name and the upper border of table.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.17. Overlapping Objects

Display parameters of overlapping objects for both new and existing graphic documents are set in **Overlapping Objects Display** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New Documents— Graphic Document— Overlapping Objects.**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Overlapping objects.**

This dialog allows to toggle on or off the background erasure: automatic interruption of break in hatches and lines when those intersect the following objects:

- ▼ text on drawing,
- ▼ arrows and/or text within:
 - ▼ dimensions,
 - ▼ Surface finish symbols
 - ▼ datum symbol,
 - ▼ leaders,
 - ▼ Stamping Symbols
 - ▼ marking symbols,
 - ▼ modification symbols,
 - ▼ reference number symbols,
 - ▼ geometric tolerance,
 - ▼ Cut Line Symbols
 - ▼ View Arrow Symbols,
 - ▼ designations of enlarged details.

If interrupt is enabled, you can:

- ▼ Activate the option of ignoring texts consisting of spaces only, i.e., cancel the clearing background under the texts containing only spaces
- ▼ Set a gap value, i.e., specify the width of a field around the stated objects provided the overlapping objects will not be displayed inside this field.

The specified gap value does not affect the extents of the erased field around text. Those extents are defined by the system automatically. They are proportional to the height of the text symbols.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

The settings made in this dialog will be used by default. If necessary, you can toggle on or off clearing the background for any of the above listed objects, by calling the same-name command from the context menu or from the **Service** menu.

9.2.6.18. Document parameters

9.2.6.18.1. View

When most of associated views display the same set of objects and use the same display parameters, then it will may be inefficient to set up objects, dimensions, process symbols and lines individually for each view. In this case, you can set up the current document, as well as all future documents, so that in their associated views the desired objects were displayed by default. This is done using **Parameters of the new view** dialog called by:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Document Parameters — View**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — Document Parameters — View**.

Tab. 9.2.47. Control elements of the views options setup dialog

Element	Setting description
Parameters Tab	Serves to set up parameters for drawing views creation. After a view is created, the display of components is controlled by Show and Hide commands from context menus of these components in the Drawing Tree.
Scale	In the provided fields, enter the scale that will be used by default used to create Arbitrary and Standard associative views, as well as when creating any non-associative views. The Scale can be defined as a ratio of any numbers. You can also select a view scale from a standard set. To do this, click the 'Zoom to Selection' button located between the zone input fields. In the upcoming menu that appears, select the desired scale.
Create New View	This option is present only in the new drawing settings dialog. Activating it means, that when creating a new drawing, the new view with the specified scale will be created in it automatically. This view will be assigned the number <i>1</i> , and the name— " <i>View 1</i> ". The coordinates origin will be positioned in the point 0.0 of the absolute coordinate system, and the view rotation angle is 0 degrees. The scale of the created view will be passed to the appropriate cell of the new drawing's title block. Regardless of the Create New View option state, when creating a new drawing, a view automatically generates its new system view in it, called Hand over.

Tab. 9.2.47. Control elements of the views options setup dialog

Element	Setting description
Create a Link to the Scaled View Scale in the Title Block	<p>Enable this option to insert a link to the scale of the first user-created view into the <i>Scale</i> cell of the title block.</p> <p>The option is operative, provided that a drawing has no other views besides the system one.</p> <p>You may create the link manually or correct the existing one by specifying another scale source view, if necessary.</p>
Transferred Components	<p>This group of options serves to specify whether or not the hidden components and the library components should be displayed by default in the following associative views:</p> <ul style="list-style-type: none"> ▼ Standard ▼ Arbitrary, ▼ Projection, ▼ View in Arrow Direction, ▼ Cut/Section *. <p>Other associative views display by default the same components as in their basic views.</p> <p>Components display management in an already created associative view is provided by the Hide and Show commands from the context menu of those components in the Drawing Tree.</p>
Projection Method	<p>The group of options allows selecting the projection method. Projection method determines the layout of views created by the Standard Views command.</p> <ul style="list-style-type: none"> ▼ If the Projection by first angle option is enabled, the view layout shown in Fig. 3.5.29, a is used. ▼ If the Projection by third angle option is enabled, the view layout shown in Fig. 3.5.29, b is used. <p>Projection method selected also affects the way the model is displayed in the view generated using the Projection View command.</p>
Lines tab	<p>Serves to set up the display of drawing the associative views of a drawing.</p> <p>Thereafter, when setting up parameters of a particular associative view, the line drawing parameters can be changed on the Lines tab of the Property bar.</p>
Visible Lines, Hidden lines, Transition Lines, Bend lines	<p>These buttons serve to define the default style of displaying the respective lines in an associative view. The line sample is displayed in an entry to the right of the button.</p>

Tab. 9.2.47. Control elements of the views options setup dialog

Element	Setting description
Transfer to the drawing	<p>This option controls the transfer of hidden contour lines to the drawing. If this option is enabled, then, when generating a new associative view, the hidden contour lines will be included in it by default.</p> <p>If Transfer to the drawing option is disabled, then the view will be generated without the hidden contour lines by default. The Show option in the Hidden Lines group will be inaccessible in this case: since there are no hidden lines, then it is not possible to show them.</p> <p>It is recommended to disable the transfer of hidden lines to the drawing when building associative views of assemblies with more than a 1000 components.</p>
Show	This option controls the display of the respective lines in an associative view. This property is available for all lines except visible ones.
Tab Objects	<p>Allows you to set the default display in associated views of model objects.</p> <p>You may further show and hide objects on the Objects page of the Property bar when customizing settings of a particular view.</p>
Transferred objects	<p>The list of objects, which images of which can be transferred from a model to drawing.</p> <p>Enable options near names of the objects which must be displayed on a drawing by default when you create the Arbitrary and Standard associated views.</p> <p>When creating other associative views, they will acquire the objects that were displayed in their base views.</p> <p>After creating a view, you can change the set of objects transferred to it.</p>
All Objects	The option allowing you to enable or disable all options of the list of objects immediately. If not all list options are enabled, then the All option is grayed out. Clicking the option in this status will disable it, the next click will enable it.
Show Hidden Objects	The option controlling display of the model hidden objects in associated views of objects that are marked as hidden in the model.
Tab Layout elements	<p>Allows you to set default display of model dimensions and process symbols in associated views— display images of threads, dimensions and designations.</p> <p>You may further show and hide dimensions and process symbols on the Objects page of the Property bar when customizing settings of a particular view. In addition, you may hide some designations.</p>

Tab. 9.2.47. Control elements of the views options setup dialog

Element	Setting description
Transferred Designations	<p>The list of dimensions and process symbols which can be transferred from a model to a drawing.</p> <p>Enable the options near the names of the designations to be transferred to a drawing by default when you create associated views.</p> <p>After creating a view, you can change the set of designations transferred to it.</p> <p>Note that dimensions and designations transferred to a drawing can be automatically hidden, if they are hidden in a model or there is at least one visible instance of this object in a drawing.</p>
All Designations	<p>The option allowing you to enable or disable all options of the list of designations immediately. If not all list options are enabled, then the All Designations option is grayed out. Clicking the option in this status will disable it, the next click will enable it.</p>

* The set of components displayed on a **Projection view**, **View by Arrow**, or on a **Cut/section** can be different from the set of components in the base view if the setting for transferring components was changed after creating the base view or if the display of some components was enabled or disabled in the assembly.

Upon finishing new view settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.



Settings made when you customize the current drawing will apply to newly created views only; parameters of the views created before the changes to the settings will stay unchanged.

9.2.6.18.2. Title block

Synchronization parameters of model data and the title block of the drawing containing associated views for both new and existing graphic documents are set in **Synchronization between title block and the model** dialog activated by: The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New Documents— Graphic Document — Document Parameters — Main text**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — Document Parameters — Main text**.

Customization of mass units carried out in the dialog covers also the nonassociative drawings. The selected measurement unit is considered at mass value transfer between the BOM item and the drawing connected to it: if different mass units are used in BOM and the drawing, then its value will be recalculated. Selection of mass unit for BOM is carried out in the value display setup dialog (see Section 9.2.5.1.4 on p. 1757).

Tab. 9.2.48. Control elements of the title block editing options setup dialog

Element	Setting description
Synchronize Title Block	This option allows to enable the data synchronization between the title block of the active drawing and that of the model (see Section 3.6.3.8.1 on p. 1232). If the option is disabled, then data synchronization does not occur. Regardless of this option state, when the first associated model view is inserted in the drawing, the drawing title block is filled with data taken from that model.
Auto	This option allows to enable auto data synchronization between the title block of the active drawing and that of the model. Automatic synchronization is run when opening and activating a document, also when rebuilding a drawing image.
upon request	This option allows to synchronize data after calling the Synchronize Data command from the title block context menu.
Synchronize Mass	Activating this option means that any time when synchronizing, the model mass will be calculated anew and recorded in the “ <i>Mass</i> ” entry.
Decimal Places	In this field you can specify, up to what digits to round the calculated mass of the model.
Mass Units	The group of options allows selection of mass units: kilograms , grams or tons . In the input field next to the measurement units, serves for you to input an additional unit name. This text is added to the mass value in the corresponding column of the title block.

Upon finishing the settings, click the **OK** button.

A change of decimal places or measurement units for the current drawing will be reflected in the drawing title block.

To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.18.3. Sheet Numbering

Setting up a drawing's sheet (only the current one) numbering is done in the **Sheet Numbering** dialog called with **Service— Parameters...** command. — **Current Drawing — Document Parameters — Sheet numbering**. Working with the dialog is described in Section 9.2.5.1.2 on p. 1755.

9.2.6.18.4. Zoning

Zoning for both new and existing drawing is turned on and set up in the dialog **Drawing Zoning** invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Document Parameters — Zoning**.

▼ for the current document — **Service — Parameters... — Current Drawing — Document Parameters — Zoning.**

This serves to enable the zoning of a drawing and to set zone dimensions and their designations.

Tab. 9.2.49. Control elements of drawing zoning settings dialog

Element	Setting description
Enable Zoning	This option allows to activate Drawing Zoning and Split Parameters settings. By default, the option is disabled, and all dialog interface items are inaccessible.
Zoning Rectangle...	The button allows to call the dialog for defining zone dimensions. In the appearing dialog, enter the zone height and width values in the respective fields or select these values from the drop-down lists. When breaking up a sheet into zones, the orientation of the dividing rectangle is ignored: the length of a sheet side is divided by the length of a rectangle size that fits the maximum number of times in the side length.
Marks	This group of elements control zone designation symbols placement on a drawing page and setting up its parameters.
Use	This group of options allows to put zone designations on a drawing sheet vertically and/or horizontally . If this option is enabled, interface items to change zone designations parameters are available.
Text type	The drop-down list allows you to choose Zone margin symbols (Western or Roman numerals, Cyrillic or Latin characters, national or English alphabet).
Start from	This field allows to you to input or set (with the counter) a digit or a number of the letter from which to start the set of designations.
Direction	The group of options allows to select a designation specifying direction: <ul style="list-style-type: none"> ▼ Bottom Up ▼ Top Down ▼ Left to Right ▼ Right to Left
Text Always Horizontal	This option allows to have all zone margin symbols positioned horizontally.
Font...	This button allows to set defining zone title font parameters in standard font setup dialog.
Line Type	This drop-down list allows to select the line type for drawing zone borders.

Tab. 9.2.49. Control elements of drawing zoning settings dialog

Element	Setting description
Width of Zone Margin, mm	The drop-down list allows to select the width of zone margin (in millimeters).
Place Outside Internal Page Boundary	This option allows to have the zone margin positioned outside of the inner page frame. If this option is disabled, then the zone margin will be positioned inside the inner page frame.

After setup is completed, click the **OK** button. The zoning parameters of the drawing will change according to the settings made.

To leave the dialog without saving changes, click the **Cancel** button.

The zoning parameters are stored in the drawing itself and do not change when the drawing is moved to another workstation.

9.2.6.18.5. Specifications — Text

Text parameters (for characters and paragraphs) of drawing specifications for both new and existing drawing are set in **Specification text parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New Documents— Graphic Document — Document Parameters — Specifications — Text**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — Document Parameters — Specifications — Text**.

Work with dialog is described in Section 9.2.4.3 on p. 1752.

9.2.6.18.6. Specifications — Parameters

Text parameters of drawing specifications for both new and existing drawing are set in **Specification parameters** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New Documents— Graphic Document — Document Parameters — Specifications — Parameters**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — Document Parameters — Specifications — Parameters**.

The dialog contains the following options:

- ▼ The option **Set Numeration When Input** controls auto-numbering of the specifications paragraphs on entry. If checked, the paragraphs are numbered automatically. Enabling or disabling this option does not affect specifications already existing in the current drawing.
- ▼ Option **Place on Last Document Pages** manages the default location of the specifications.

When the **Place on Last Document Pages** option is enabled, the specifications page is automatically created on the last document page. If the volume of the technical specifications text exceeds the page size, then a new page will be created on the previous sheet. The specifications text is distributed between pages as follows.

- ▼ - The start of specifications moves to the page of the previous sheet, which it fills entirely.

- ▼ - The text extending beyond this page borders is placed on the specifications sheet located on the last page.

The newly entered specifications will be filling this page. If, as you keep in putting in specifications, their volume exceeds the page size again, the next page will be automatically created on the previous page, and the specifications text will be redistributed between pages similar to the described above. Thus, as specifications are entered, their beginning will move to the previous drawing pages.

If the **Place on Last Document Pages** option is disabled, the specifications page is created on the first drawing page. If the volume of the technical specifications text exceeds the page size, a new page will be created automatically to the left of the first page frame. The start of specifications remains on the first drawing sheet in this case. Upon the continued with the next input of specifications, those will be moved left from the first drawing page.

This checkbox state is determined by the execution of the **Auto Locate Specifications** command. This checkbox affects the location of the specifications only if it was changed manually (see Section 3.5.7.2 on p. 1200).

- ▼ The **Synchronize technical specifications** option controls automatic transfer of technical specifications from a model to drawing. If it is enabled, technical specifications set in the model are transferred to technical specifications during creation of the associative view; if the option is disabled, technical specifications are not transferred. Conditions of transferring technical specifications from a model to drawing and the methods of their update after editing are described in Section 3.6.3.9 on p. 1234.

Upon finishing Specification Options settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

The specifications settings are stored in the drawing itself and does not change when the drawing is moved to another workstation.

9.2.6.18.7. Unspecified Surface Finish

Text parameters of the unspecified surface finish symbol cannot be entered during the symbol creation. These parameters are defined by the document's setup (both for new and existing documents). This setting is done in the **Unspecified Surface Finish Parameters** dialog activated by

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Document Parameters — Unspecified surface finish,**
- ▼ for the current document — **Service — Parameters... — Current Drawing — Document Parameters — Unspecified surface finish.**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

After finishing setting up the parameters and closing the parameters dialog, the text for unspecified surface finish symbol will change according to the new settings.

The unspecified surface finish text parameters are stored in the drawing itself and do not change when the drawing is moved to another workstation.

9.2.6.18.8. BOM name on a sheet

Parameters of BOM name on a sheet for both new and existing drawings are set in **Parameters of BOM name on a sheet** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Document Parameters — BOM name on the sheet,**
- ▼ for the current document — **Service — Parameters... — Current Drawing — Document Parameters — BOM name on the sheet.**

The dialog allows you to customize default settings of BOM name text (see Section 6.2.6.12.3 on p. 1532) and the distance from the BOM table to the name.

Tab. 9.2.50. Control elements of the BOM name on a sheet setup dialog.

Element	Setting description
Line spacing, mm	The field allows to set the distance between text lines.
First line Indent, mm	This field allows to set the value of the indentation to the right from the start of a paragraph in its first line.
Font...	A button to call the font parameter setup dialog.
Tabs...	A button to call the dialog to set up tabulation parameters.
Indentations	This group of fields allow to set up values of left and right indentations of paragraph (in millimeters). BOM name paragraph borders are vertical lines on which the right and left borders of BOM table lie. These lines are used to mark off the left indents, right indents and first line indent. In other words, the maximum name paragraph width is equal to the BOM table width.
Spacing	This group of fields allow to set up values of spacing before and after the paragraph (in millimeters).
Alignment	This group of fields allows to define the desired paragraph alignment method (left, right, centered, justified).
Distance to BOM, mm	This field allows to set up the required distance between BOM name and the upper border of BOM table.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.19. First Sheet Parameters

9.2.6.19.1. Format

You can choose a first sheet format for both new and existing graphic documents from a standard set or to define arbitrary sheet dimensions. This is done using **Sheet format** dialog called one of the following ways:

- ▼ for new documents— **Service— Parameters...— New documents— Graphic document— First sheet parameters— Format**,
- ▼ for current document, **Service— Parameters...— Current Drawing— First Sheet parameters— Format** or calling the **Format** command from the context menu of the List of Sheets, Views and Layers in the **Document Manager** dialog (see Section 3.5.6.2 on p. 1182).
Work with **Sheet format** dialog is described in Section 9.1.12.3.1 on p. 1743.

9.2.6.19.2. Template

First sheet template for both new and existing graphic documents is set in **Template** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Sheet parameters — Layout**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — First sheet parameters — Layout**.

Working with the dialog is described in Section 9.2.4.2.2 on p. 1750.

9.2.6.19.3. Revision Table

Revision table parameters for the first sheet of both new and existing graphic documents are set in **Revision table** dialog activated by:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — First sheet parameters — Table of changes**,
- ▼ for the current document — **Service — Parameters... — Current Drawing — First sheet parameters — Table of changes**.

The similar dialog is used when setting up revision tables for new sheets of graphic documents.

The dialog contains one option— **Split Into Blocks**. When the option is disabled, all objects of the revision table are put one above another, making up a single block. Use this option to split the revision table into blocks.



The size (number of lines) of these is defined when setting up the layout style of the current document.

After setup is completed, click the **OK** button.

When setting up the current document, the changes will be displayed on the screen. When setting up new documents, the changes will be in effect at creation.

To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.20. Parameters of new sheets

9.2.6.20.1. Format

You may choose the format of new sheets of graphic documents (both new and current) from the standard set or to define arbitrary dimensions of sheets. To do this, use the **Sheet Format** dialog invoked with the following command:

- ▼ for new documents:— **Service— Parameters— New documents— Graphic document— Parameters of new sheets— Format,**
- ▼ for current document:— **Service— Parameters— Current drawing— Parameters of new sheets— Format.**

Working with the dialog is described in Section 9.1.12.3.1 on p. 1743.

9.2.6.20.2. Layout

You may choose the layout of new sheets of graphic documents (both new and current) in the **Layout** dialog invoked with the following command:

- ▼ for new documents:— **Service— Parameters— New documents— Graphic document— Parameters of new sheets— Layout,**
- ▼ for current document:— **Service— Parameters— Current drawing— Parameters of new sheets— Layout.**

Working with the dialog is described in Section 9.2.4.2.2 on p. 1750.

9.2.6.20.3. Revision Table

Revision tables for new sheets of graphic documents (both new and current) are set up in the **Revision table** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — New sheets' parameters — Table of changes,**
- ▼ for the current document — **Service — Parameters... — Current Drawing — New sheets' parameters — Table of changes.**

Working with the dialog is described in Section 9.2.6.19.3 on p. 1829.

9.2.6.21. Parametrization

The use of the parametric mode of the KOMPAS-3D system for graphic documents (both new and current) is set up in the **Parametrization management** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Graphic Document — Parametrization,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Parametrization.**

A similar dialog is used for management of parametrization when working with sketches.

Tab. 9.2.51. Control elements of the parametric mode setup dialog

Element	Setting description
Associate at entry:	<p>The list contains names of object types. Activate the options next to the object type names which should be created as associated (related with other objects) when constructed. When associated dimensions are activated, the associated diameter, radius and angular dimensions will be applied only to curves that support associativity. Associated dimensions cannot be applied to arcs and line segments in a contour or macro element, to sides of rectangles and polygons.</p>
Parametrize:	<p>The list contains names of types of relations between object types. Activate the options next to the names of type of relations between objects that need to be automatically constrained upon creating and editing drawn images.</p>
All	<p>Use this option to activate or deactivate association of all objects (parameterization of all types of relations). If association is enabled not for all objects (parameterization is enabled not for all types of relations), then the option is displayed on a gray background. Clicking the option in this status will disable it, the next click will enable it.</p>
Fix dimensions	<p>Enabling this option means that associative dimensions during creation will be automatically fixed with and bound to a variable. If the option is disabled, associative dimensions will be created as informational and without variables. For more information about controlling and information dimensions, see Section 7.2.1.10 on p. 1606. This option is accessible if the Dimensions option in the group Associate at Input is enabled.</p>
Fix the length of auto center line	<p>Enabling this option means that if at least one point of the auto centerline is free from constraints, the restriction of <i>fixed length</i> will be additionally applied. If this option is disabled, the restriction is not applied. See more about restrictions automatically applied to auto centerlines in Section 7.2.6.5 on p. 1627.</p>
Disable all	<p>This option allows to enable and disable parametrization mode in a document.</p> <ul style="list-style-type: none"> ▼ If this option is disabled, the parametrization mode is enabled and the options to setup this mode are accessible in this dialog. ▼ If this option is disabled, the parametrization mode is enabled and the options to setup this mode are not accessible. <p>Enabling and disabling the parametrization mode does not change its setup. You can also enable or disable the parametric mode using button Parametrization Mode on the Current Status panel.</p>



Enabling the parametrization mode will have no effect if all options are disabled in the groups **Associate at Input:** and **Parametrize:**. In this case, no relations and restrictions will be applied, i.e. patterning will be carried out in the mode as if the parametrization mode had been disabled.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

The settings of the parametric mode are saved in the document itself and are not modified when transferring the document to the other work place.

9.2.6.22. Numbering

In order to setup object numbering parameters in numbering groups and to add/change/delete user groups the following dialogs are used: **System numbering group parameters** and **User numbering group parameters**.

The dialogs are called by the following commands: **Service — Parameters... — Current drawing/fragment — Numbering — System groups** and **Service — Parameters... — Current drawing/fragment — Numbering — User groups, respectively**.



For quick calling the dialogs, the **Numbering group parameters** button is used. It is located on the toolbar of the **Numbering** window or on the Special control panel when working with the **Change sign** command.

Numbering system group parameters dialog

The dialog contains the **#Change designation numbering** system group and the list of its related numbering parameters. Values of these parameters may be changed by manual setting or by selection from predefined lists. The list of parameters and the order of setting their values can be found in Table 9.2.52.

Tab. 9.2.52. Numbering parameters

Name	Setting description
Type of ?number	<p>The numbers of objects in a group may be represented by numbers from 1 to 2 147 483 647 or by custom symbols.</p> <p>By default, number are used for numbering. The parameter's value is as follows: 1,2,3,4,5...</p> <p>In order to use symbols for setting numbers, click in the parameter's value field and choose the Set symbols option from the drop-down list. The list of symbols in use is set in the Set of symbols field.</p>

Tab. 9.2.52. Numbering parameters

Name	Setting description
Number length	<p>The field is present in the list in case of numeric object numbering (the Number type parameters has the following value: 1,2,3,4,5...).</p> <p>Defines the minimal number of digits in the object's number.</p> <p>The number of digits may be set as a natural number between 1 and 10.</p> <p>If the number of digits in the object's number is less than the set one, zeros are added before the number.</p> <p>If the number of digits in the object's number is more that the set one, the number remains unchanged.</p>
Set of characters	<p>The field is present in the list in case of symbol object numbering (the Number type parameters has the following value: Set symbols).</p> <p>It defines which symbols and in what order will be assigned to objects that are being added to the group of numbering.</p> <p>By default, the symbol set consists of Russian letters. You can add custom symbols to the set. Duplicate symbols are not allowed. The maximum total number of symbols in the set is 64.</p> <p>The number of objects in a group may be more than the number of set symbols. In this case, set symbols are used first for object numbering. Then, the same symbols with numbers are used. For example: <i>A1, B1, C1, ..., A2, B2, C2</i> and so on.</p> <p>Note that when switching from symbol numbering to numeric numbering, the set of symbols is reset to the default one.</p>
Text before number	<p>You can use it to enter a string of symbols which will be displayed before the object number. The maximum number of symbols in a string is 256.</p>
Text after number	<p>You can use it to enter a string of symbols which will be displayed after the object number. The maximum number of symbols in a string is 256.</p>
Start from	<p>You can enter a number or a symbol's serial number (in case of symbol numbering) from which object numbering will begin in the group.</p> <p>For example, the set of numbering symbols consists of Russian letters <i>A, B, B</i> and so on. Enter 3, in the <i>Start from</i> field. The object numbering in the group will start from letter <i>B</i>.</p> <p>if the value of the Start from parameter is changed, the group's object numbers will be changed automatically. The exception is the case when numbering auto determination is disabled (the Auto-rebuilding of numbering parameter has value No) and the number is being decreased. Object numbers will not change in this case. Empty strings with free numbers will appear in the list before the first object. The first empty string in the list contains the first number.</p>

Tab. 9.2.52. Numbering parameters

Name	Setting description
Auto-rebuilding of numbering	<p>Use it to manage automatic rebuilding of group objects' numbers. Click in the parameter's value field and choose the required value from the dropdown list.</p> <p>If the parameter has value Yes, the numbering will be started from the first number (this number is set in the Start from field). If an object is deleted from the object numbering group, those positioned after the deleted one will be changed automatically.</p> <p>If the parameter has value No, the numbering may start from a custom number which should be greater than the starting one. Group objects' numbers do not change in case of an object's deletion. The strings containing free numbers will remain in the list.</p> <p>Note that when switching to the string's Yes value, the strings with free numbers will be deleted automatically.</p>

If necessary, you can return to default numbering parameters' values. To do so, select the name of the group click the **Reset** button.

Numbering user group parameters dialog

You can use it to create numbering groups and set up numbering parameters for each of them. By default, the dialog contains no user groups.

To create a group, click the **Create** button. Default name and parameters of the created group will be displayed in the dialog's list of groups. The dialog for setting the group's name will appear. You can use it to change the default name. Make changes and click the **OK** button. The **Cancel** button closes the dialog without changing the group's name. The group's name must be unique within the current document. Its length should not be more than 256 symbols.

In future, you will be able to change the group's name or delete the group.

In order to change the name, select it and click the <F2> button. The dialog of the group name setting will appear on the screen. Make the required changes and click the **OK** button. The **Cancel** button closes the dialog without changing the group's name.

To delete a group, select it and click the **Delete** button. If a group contains objects, the request to confirm deletion will appear on the screen. If a nonempty group is being deleted, the links to object numbers contained in the group will become invalid.

The same set of numbering parameters corresponds to the user group as well as to the system group. The list of parameters and the order of setting their values can be found in Table 9.2.52.



When a group is created, its parameters have default values. To make a new group have the same numbering parameters as an existing one, select the name of the group before creating the new one.

After setup is completed, click the **OK** button. Changes of numbering groups will be transferred to the list of groups displayed in the **Numbering** window.



The list of numbering groups is displayed not only in the **Numbering** window, but also in the Property bar when using commands for creating numbered objects (see Section 3.9.1.1 on p. 1277).

The setup dialog can also be called while the **Change sign** command is in operation. In this event, instead of being transferred to the list right after the dialog closes, changes inside groups will be applied only in numbering parameters setup for already listed groups. In order to bring the list in accordance with all changes made, close the **Change sign** command and call it again.

To leave the dialog without saving changes, click the **Cancel** button.

9.2.6.23. Report table parameters

9.2.6.23.1. Title

Parameters of text for table title in a report for graphic documents (both new and current) are set up in the **Parameters of text for table title** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Report table parameters — Heading,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Report table parameters — Heading.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

The settings are used when posting a report in a graphic document and when displaying a graphic document report in the Data Processing Window.

9.2.6.23.2. Cell

Parameters of text for table cell in a report for graphic documents (both new and current) are set up in the **Parameters of text for table cell** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Report table parameters — Cell,**
- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Report table parameters — Cell.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

The settings are used when posting a report in a graphic document and when displaying a graphic document report in the Data Processing Window.

9.2.6.23.3. Table name

Parameters of the name of the associative report's table on a sheet for graphical documents can be set up in the **Table name text parameters** dialog called by the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Graphic document — Report table parameters — Table name,**

- ▼ for the current document — **Service — Parameters... — Current drawing/fragment — Report table parameters — Table name.**

The dialog is used to set up default text parameters of the table name for an associative report and the distance between the table and the name (see Section 4.2.2.1.3 on p. 1364 on table names).

Work with dialog is described in Section 9.2.6.16.3 on p. 1818.

Names of non-associative reports can be set up for names of simple tables (see the Section referred to above).

9.2.7. Model

9.2.7.1. Default font

The font that will be used by default in models (both new and current) is carried out in the **Default font** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Model — Default font,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly/Current sketch — Default font**

Working with the dialog is described in Section 9.2.4.1 on p. 1749.

9.2.7.2. Dimensions

9.2.7.2.1. General settings

General settings of dimensions for models (both new and current) are implemented in the **General settings** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — General settings,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — General settings.**

Working with the dialog is described in Section 9.2.6.10.1 on p. 1771.

9.2.7.2.2. Parameters

Displaying of dimensions for models (both new and current) is set in the **Parameters for displaying of dimensions** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly/Current sketch — Dimensions — Parameters.**

Working with the dialog is described in Section 9.2.6.10.2 on p. 1772.

9.2.7.2.3. Arrows and obliques

Drawing of arrows and obliques for models (both new and current) is set in the **Arrows and obliques** dialog which is called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Arrows and obliques,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — Arrows and obliques.**

Working with the dialog is described in Section 9.2.6.10.4 on p. 1773.

9.2.7.2.4. Arrow filter— Linear dimensions

The filter of arrows of linear dimensions for models (both new and current) is set up in the **Filter of arrows of linear dimensions** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Arrow filters — Linear dimensions,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — Dimensions — Arrow filters — Linear dimensions,**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.7.2.5. Arrow filter— Dimensions of circles and arcs

The filter of arrows of dimensions of circles and arcs for models (both new and current) is set up in the **Filter of arrows of linear dimensions** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Arrow filters — Dimensions of circles and arcs,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — Dimensions — Arrow filters — Dimensions of circles and arcs,**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.7.2.6. Arrow filter— Angular dimensions

The filter of arrows of angular dimensions for models (both new and current) is set up in the **Filter of arrows of angular dimensions** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Arrow filters — Angular dimensions,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — Dimensions — Arrow filters — Angular dimensions,**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.7.2.7. Dimension text

Parameters of dimension text for models (both new and current) are set up in the **Parameters of dimension text** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Text,**

- ▼ for the current document — **Service — Parameters... — Current part/assembly/Current sketch — Dimensions — Text.**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.7.2.8. Dimension text location

The location of dimension texts for models (both new and current) is set up in the **Parameters of dimension text caption** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Text's position,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — Text's position.**

Working with the dialog is described in Section 9.2.6.10.9 on p. 1778.

9.2.7.2.9. Tolerances and tolerance range — Parameters

Display of tolerances and tolerance range of dimensions for models (both new and current) is set up in the **Parameters of tolerances and tolerance range** dialog called with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Model — Dimensions — Tolerances and tolerance range — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — Tolerances and tolerance range — Parameters.**

Working with the dialog is described in Section 9.2.6.10.10 on p. 1779.

9.2.7.2.10. Tolerances and tolerance range — Default tolerances

Default parameters of tolerances for models (both new and current ones) are set up in the **Value and state of default tolerances** dialog called by this command:

- ▼ for new documents — **Service — Parameters... — New documents — Model — Dimensions — Tolerances and tolerance range — Default tolerances,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Dimensions — Tolerances and tolerance range — Default tolerances.**

The dialog contains the following controls:

- ▼ A group of buttons for **Default tolerances of dimensions**. The buttons are used to call the **Tolerance** dialog where the default value of tolerance is set either by specifying the class of accuracy or entering maximum deviations. Setup is performed separately for each type of dimensions: linear, angular, diameter, radial (only deviations are set up for angular dimensions). Setup parameters belong to subgroup 2.1 (see Section 9.2.1.1 on p. 1747).

There is a field next to each button for previewing the current value of tolerance. The field cannot be edited manually; content variants:

- ▼ 10 with a class of accuracy or deviations¹ — dimensions of this type will have the specified individual tolerance,

1. 10 is for illustration purposes.

- ▼ an empty field — neither the class of accuracy nor deviations are set up, dimensions of this type will have a general tolerance.



The field can only contain 10. This means the tolerance is set by a class of accuracy but the number of the class of accuracy for which display in the dimension text is disabled has been selected. The accuracy class limiting number is specified during the setup of precision of dimension texts (see Section 9.2.7.2.11 on p. 1839).

- ▼ The group of options **Include tolerances in dimensions** allows selecting objects whose dimensions (parameters) tolerances are automatically assigned to. When the option is disabled, dimensions (parameters) of respective objects will have no tolerance by default — neither a general nor an individual one.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

More on tolerances in a model, see Section 2.10.2 on p. 667.

9.2.7.2.11. Precision

Parameters of displaying values of dimension text for models (both new and current) are set up in the **Accuracies of dimension text** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Dimensions — Accuracies**,
- ▼ for the current document — **Service — Parameters... — Current part/assembly/Current sketch — Dimensions — Accuracies**.

Working with the dialog is described in Section 9.2.6.10.12 on p. 1781.

9.2.7.3. Reference marks

9.2.7.3.1. Leader — Parameters

Parameters for displaying of leaders for models (both new and current) are set up in the **Parameters of displaying leaders** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Leader — Parameters**,
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Leader — Parameters**,

Working with the dialog is described in Section 9.2.6.11.1 on p. 1782.

9.2.7.3.2. Leader — Arrows and obliques

Drawing of leaders' arrows and obliques for models (both new and current) is set in the **Arrows and obliques** dialog which is called by the command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Leader line — Arrows and obliques**,

- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Leader — Arrows and obliques.**

Working with the dialog is described in Section 9.2.6.11.2 on p. 1783.

9.2.7.3.3. Leader — Arrow filter

The filter of arrows of leaders for models (both new and current) is set up in the **Filter of arrows of leaders** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Leader line — Arrow filter,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Leader line — Arrow filter,**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.7.3.4. Leader— Text above/below/behind shelf

Parameters of text on leaders for models (both new and current) are set up in the **Parameters of text above/below/behind shelf** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Leader line — Text above/under/behind the shelf,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Leader line — Text above/under/behind the shelf,**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.7.3.5. Leader — Inclined text

Parameters of inclined text on leaders for models (both new and current) are set up in the **Parameters of inclined text** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Leader line — Inclined text,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Leader line — Inclined text,**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.7.3.6. Leader— Symbol sizes

Dimensions of sizes of symbols for models (both new and current) are set up in the **Symbol sizes** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Leader line — Symbol sizes,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Leader line — Symbol sizes,**

Working with the dialog is described in Section 9.2.6.11.6 on p. 1784.

9.2.7.3.7. Designation of position— Parameters

Parameters for displaying of designations of positions for models (both new and current) are set up in the **Parameters of displaying position designations** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Designation of position — Parameters,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Designation of position — Parameters,**

Working with the dialog is described in Section 9.2.6.13.1 on p. 1787.

9.2.7.3.8. Reference number— Parameters

Parameters of form for designations of positions for models (both new and current) are set up in the **Parameters of form for position designations** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Designation of position — Form parameters,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Designation of position — Form parameters,**

Working with the dialog is described in Section 9.2.6.13.2 on p. 1787.

9.2.7.3.9. Designation of position— Arrows and obliques

Parameters for displaying arrows of designation of positions for graphic documents (both new and current) are set up in the **Arrows** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Designation of position — Arrows and obliques,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Designation of position — Arrows and obliques,**

Working with the dialog is described in Section 9.2.6.13.3 on p. 1789.

9.2.7.3.10. Designation of position— Arrow filter

Parameters for the filter of arrows of position designation for models (both new and current) are set up in the **Filter of arrows of position designations** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Designation of position — Arrow filter,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Designation of position — Arrow filter,**

Work with dialog is described in Section 9.2.6.10.5 on p. 1774.

9.2.7.3.11. Designation of position— Text for designation of position

Parameters of text for position designations for models (both new and current) are set up in the **Parameters of text for position designations** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Designation of position — Position designation text,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Designation of position — Position designation text,**

Work with dialog is described in Section 9.2.6.10.8 on p. 1777.

9.2.7.3.12. Surface finish

Parameters for designation of surface finish for models (both new and current) are set up in the **Parameters for designation of surface finish** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Surface finish,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Surface finish,**

Work with dialog is described in Section 9.2.6.13.6 on p. 1790.

9.2.7.3.13. Shape deviations and datum

The parameters of shape deviation and datum drawing for models (both new and current) are set up in the **Parameters of shape deviations and datum** dialog called by the :

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Shape deviations and datum,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Leader — Shape deviations and datum.**

Work with dialog is described in Section 9.2.6.13.7 on p. 1791.

9.2.7.3.14. Auto-sorting

The auto-sorting mode for models (both new and current) is enabled and set up in the **Auto-sorting of letter symbols** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Designations — Auto sorting,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Designations — Auto sorting,**

Working with the dialog is described in Section 9.2.6.13.14 on p. 1796.

9.2.7.4. Part

9.2.7.4.1. Properties




New parts' properties are set in the **Part's properties dialog which is called by the Service — Parameters... — New Documents — Model — Part — Properties command**,

The dialog makes it possible to enter parameters that will be used by default as properties — name and designation of the part, material and density of the material, which the part is made of, hatch parameters for displaying the part in cuts and sections on the model's associative drawings.

Tab. 9.2.53. Control elements of the Properties Setup dialog

Element	Setting description
Designation	The field allows to enter the designation which will be assigned to a part by default.
Name	The field allows to enter the name which will be assigned to a part by default.
Material	<p>The group of elements makes it possible to set the default material and its density, parameters of the part's hatching. The table contains values of properties that can be adjusted.</p> <ul style="list-style-type: none"> ▼ Choose the material from the list or from the Materials and Assortments Library using the Select material from list or Select material from reference button. Once material is selected, cells of the Value column are filled automatically. ▼ If you need to manually enter density or hatch parameters, click the required cell and specify the value.
Material designation	<p>Part's material designation field. It is filled automatically after selection of the material. This field is not available for manual entry. You can delete the material's density using the Delete button. The material's density is transferred to the drawing's title block when the part's associative view is created in it (if this view is the first in the order of their insertion into the drawing).</p>
Density, g/mm³	<p>Material density field. It is filled automatically after selection of the material. This field is available for manual entry. The material's density is used during calculation of a part's mass-center characteristics if calculated by density.</p>
Hatch style	<p>Hatch style field. It is filled automatically after selection of the material. The field supports choosing style from the list, as well as from the hatch style library. Hatch style is transferred to the drawing when creating a cut/section of the part's associative cut/section.</p>

Tab. 9.2.53. Control elements of the Properties Setup dialog

Element	Setting description
Hatch color	Hatch color field. By default, the color set in the hatch style is displayed. You can change color in this field. Hatch color is transferred to the drawing when creating a cut/section of the part's associative cut/section.
Hatch angle	Hatch angle field. The field supports selection of values from the list and manual entry. Hatch angle is transferred to the drawing when creating a cut/section of the part's associative cut/section.
 Select from List of Materials	You can select material from the density reference file (... \ASCON\KOMPAS-3D V... \Sys\Graphic.dns file; the values of material densities contained therein may be edited by the user). On clicking the button the material's density selection dialog will be displayed. Once material is selected, values of the following properties are filled automatically: Material's designation, Density, Hatch style, Hatch color.
 Select from Reference of Materials	This button allows to select a material from an external Materials and Assortment Library (the Reference source should be linked to KOMPAS-3D). Once material is selected, values of the following properties are filled automatically: Material's designation, Density, Hatch style, Hatch color.
 Delete symbol	The button allows to clear the Material's density field.

After properties of a part are set up, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.2. Customization of properties list

You can customize connection to necessary libraries of properties and displaying of properties (both new and current) in details. To do this, use the **Customization of properties list** dialog. This dialog is displayed in the following cases:

- ▼ for new documents — after calling the **Service — Parameters... — New Documents — Graphic Document — Property list setup** command,
- ▼ for the current document — after calling the **Service — Parameters... — Current part— Customization of properties list** command, as well as after pressing the button **Customization of property list** on the Property panel when properties of a detail or body are set.

For new documents, connection to necessary property libraries and displaying of properties in a document are customized in the dialog. For current document, only displaying of properties in a document is customized.

Working with the dialog is described in Section 9.2.6.5 on p. 1762.

9.2.7.4.3. New versions' numbers

Parameters of forming the numbers of new versions are set up in the **New versions' numbers** dialog. This dialog is displayed in the following cases:

- ▼ for new documents — after calling the **Service — Parameters... — New Documents — Graphic Document — New version's numbers command**,
- ▼ for the current document — after calling the **Service — Parameters... — Current part — New versions' numbers** command, as well as after clicking the **Parameters of forming new versions' numbers** button in the Document Manager when working with versions and additional numbers.



The same dialog is used when adjusting numbers of new versions for simple and technological assemblies.

Tab. 9.2.54. Control elements of the new versions' numbers setup dialog

Element	Setting description
Generate numbers for new versions	<p>The option makes it possible to control automatic forming of numbers when creating versions.</p> <p>If the option is enabled, the number of the first version is created automatically in accordance with the defined parameters.</p> <p>If the option is disabled, when creating a dependent version, it will be assigned the number of the source version. No number will be assigned when an independent version is created.</p>

Tab. 9.2.54. Control elements of the new versions' numbers setup dialog

Element	Setting description
Continuous numbering	<p>This option makes it possible to set the procedure of version numbers forming.</p> <ul style="list-style-type: none"> ▼ If the option is enabled for each new version, both dependent and independent, the number will be formed which follows the greatest version number in the document. ▼ If the option is disabled, the number of the version being created is formed based on the nesting level of the version and the independent version to which it is subordinate. <p>For example, the source version is <i>ABC-123.45-03</i>, where <i>ABC-123.45</i> is the base designation, while <i>03</i> is the version's number. This number is the greatest one.</p> <ul style="list-style-type: none"> ▼ If the option is enabled, the new version (dependent or independent) is assigned the number of <i>04</i>. Designation of the version has the following format: <i>ABC-123.45-04</i>. ▼ If the option is disabled, new dependent version is created with <i>03-01</i> number. Its designation has the following format: <i>ABC-123.45-03-01</i>. The new independent version is created with number <i>04</i>. The version's designation is <i>ABC-123.45-04</i>.
Separating symbol	The field makes it possible to set a symbol which will be displayed in front of the number in the version's designation.
Add Leading Zeros to Number	The option makes it possible to control adding zeros in front of the figure included in the number. If the option is enabled, zeros are added. If the option is disabled, no zeros are added. The number of added zeros depends on the number's length.
Number length	You can use the drop-down list to set the minimal number of digits in the number.
Start from	You can use this field to enter or set by counter the number from which the numbering of versions will begin.
Generate additional version numbers	<p>You can use this option to control the procedure for generation of additional numbers.</p> <ul style="list-style-type: none"> ▼ If the option is enabled, numbers are generated automatically in accordance with the settings described below. ▼ If the option is disabled, numbers are not generated. <p>If a document contains numbers generated with this option enabled, disabling the option later will not affect the order of number generation, i.e. the next number will be created with the same settings as the numbers that are present in the document.</p>

Tab. 9.2.54. Control elements of the new versions' numbers setup dialog

Element	Setting description
Separating symbol	The field makes it possible to set a symbol which will be displayed in front of the additional number in the version's designation.
Add Leading Zeros to Number	The option makes it possible to control adding zeros in front of the figure included in the additional number. If the option is enabled, zeros are added. If the option is disabled, no zeros are added. The number of added zeros depends on the additional number's length.
Number length	You can use the drop-down list to set the minimal number of digits in the additional number.
Start from	You can use this field to enter or set by counter the number from which the numbering of versions will begin.

After making the required settings, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.4. Color

Color and optical properties of new parts' surfaces are set in the **Part's color** dialog which is called by the **Service —Parameters... — New Documents — Model — Part — Properties command**,

The similar dialog is used when setting up color and optical properties for simple and technological assemblies.

Please note that components in a model preserve their own colors. Settings made in the dialog only relate to operations of adding or removal of material that are carried out in a model itself.

Tab. 9.2.55. Control elements of the dialog to setup color and optical properties

Element	Setting description
Color	The button allows to choose color of a part in the standard color selection dialog.
Optical properties	A set of controls allows to setup optical properties of surfaces of a part.
Background color	To define values of the parameters, position each slider as appropriate.
Diffusion	The numerical value of a parameter is displayed in the information field.
Specularity	
Shininess	
Transparency	
Emission	

Tab. 9.2.55. Control elements of the dialog to setup color and optical properties

Element	Setting description
Preview Window	The preview window displays a sphere with surface properties specified in the dialog. This helps to make visual evaluation of introduced changes.

After color and optical properties of a part are set up, press the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.5. Properties of the absolute coordinate system

Properties of the absolute coordinate system for parts (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — Properties of absolute CS command,**
- ▼ for the current document — **Service — Parameters... — Current part — Properties of absolute CS.**

A similar dialog is used when setting up properties of the absolute coordinate system for simple and technological assemblies.

The dialog allows to set names and color of coordinate planes, axes and symbols for the coordinate system origin.

Tab. 9.2.56. Control elements of the dialog to set up the absolute coordinate system

Element	Setting description
List of objects	This list contains objects of which name and color can be customized: three coordinate planes, three coordinate axes and the origin symbol. You may set different colors or the same colors to display these objects in the window for a model of these objects. To specify the objects to be customized, select them in the list with a click. To rename an object, select it and click on it. The line then opens for editing, and you can enter the desired name in it. After entering the name, press the <Enter> key to confirm it or click on a blank space in the dialog. Object names entered in this way will be displayed in the Design Tree.
Use Part (Assembly) Color	Enable this option if all elements of the selected type should be displayed in the part (assembly) color. Disable this option if the color of elements of the selected type should differ from the part (assembly) color.
Color	Click this button to choose a color of the selected object. The button is available in the dialog when the Use Part Color option is disabled.

Tab. 9.2.56. Control elements of the dialog to set up the absolute coordinate system

Element	Setting description
All by default	If you need to restore the color by default for all element types, click this button.
Preview Window	Changes in the settings of displaying elements of a model are shown in the preview window. This helps to make visual evaluation of introduced changes. The Preview window is available in the dialog when the Use part color option is disabled.

After parameters of the absolute coordinate system are set up, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.6. Properties of local coordinate systems

Properties of local coordinate systems for parts (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — Properties of local CS command**,
- ▼ for the current document — **Service — Parameters... — Current part — Properties of local CS**.

A similar dialog is used when setting up properties of local coordinate systems for simple and technological assemblies.

The dialog allows to set names and color of coordinate planes, axes and symbols for the coordinate system origin.

Tab. 9.2.57. Control elements of the dialog to set up local coordinate systems

Element	Setting description
List of objects	<p>This list contains objects of which name and color can be customized: three coordinate planes, three coordinate axes and designation of the local coordinate system.</p> <p>You may set different colors or the same colors to display these objects in the window for a model of these objects.</p> <p>To specify the objects to be customized, select them in the list with a click.</p> <p>To rename an object, select it and click on it. The line then opens for editing, and you can enter the desired name in it. After entering the name, press the <Enter> key to confirm it or click on a blank space in the dialog. Object names entered in this way will be displayed in the Design Tree.</p>

Tab. 9.2.57. Control elements of the dialog to set up local coordinate systems

Element	Setting description
Use Part (Assembly) Color	Enable this option if all elements of the selected type should be displayed in the part (assembly) color. Disable this option if the color of elements of the selected type should differ from the part (assembly) color.
Color	Click this button to choose a color of the selected object. The button is available in the dialog when the Use Part Color option is disabled.
All by default	If you need to restore the color by default for all element types, click this button.
Preview Window	Changes in the settings of displaying elements of a model are shown in the preview window. This helps to make visual evaluation of introduced changes. The Preview window is available in the dialog when the Use part color option is disabled.

After parameters of local coordinate systems are set up, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.7. Properties of objects

Properties of objects for parts (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — Properties of objects command**,
- ▼ for the current document — **Service — Parameters... — Current part — Properties of objects**.

A similar dialog is used when setting up properties of objects for simple and technological assemblies.

The dialog allows to set a color of displaying various objects.

Tab. 9.2.58. Control elements of the properties setup dialog

Element	Setting description	Subgroup number*
List of elements	The list contains all types of objects of a model. Next to an element name there is an icon corresponding to it in the Design Tree. To change the color of displaying elements of a specific type, select the type name.	—

Tab. 9.2.58. Control elements of the properties setup dialog

Element	Setting description	Subgroup number*
Use Part (Assembly) Color	Activate this option if all elements of the selected type should be displayed in the part (assembly) color. Disable this option if the color of the selected element type should differ from the part (assembly) color.	2.1
Color	Click the button to set the color of elements of the selected element type. The button is available in the dialog when the Use part (assembly) color option is disabled.	2.1
Advanced	Press the button to choose optical properties of surface of elements of the selected type. This button is accessible when setting up all elements except for axes, planes and sketches, if the Use part (assembly) color option is disabled.	2.1
All by default	If you need to restore the color by default for all element types, click this button.	2.1
Preview windows	Changes in the model element display settings are displayed in the preview window. This helps to make visual evaluation of introduced changes. The Preview windows is available in the dialog when the Use part color option is disabled.	—

* Subgroup number to which adjustable parameter belongs. Parameters are combined in groups and subgroups by their action to the current documents objects, see Section 9.2.1.1 on p. 1747.

When parameters of displaying objects are set, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.8. Properties of sheet body

Default parameters of sheet bodies for parts (both new and current) are set up in the **Properties of sheet object** dialog invoked with the following command:


- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — Properties of sheet body command**,
- ▼ for the current document — **Service — Parameters... — Current part — Properties of sheet body**.

Note that the settings of parameters of sheet bodies for current part differs from the setting of these parameters for new parts. The description provided in the table corresponds to the settings of parameters of sheet bodies for new parts. Specifics of implementation of setting for the current part are described below the table.

Tab. 9.2.59. Control elements of the dialog to setup parameters of a sheet body

Element	Setting description
Thickness, mm	The field to enter the sheet body thickness.
Radius, mm	The field to enter the bend radius. The Outer option allows to choose a method of creating bends. If it is enabled, then bends will be created by the outer radius, otherwise* by the inner radius.
Angle, degrees	The field to enter the default value of the bend angle. The Complimentary option allows to specify the angle interpretation. If this option is enabled, the angle is considered as the complementary angle, otherwise is considered as the bend angle (see Section 2.4.4.3.1 on p. 254).
Release	Enable this option to allow for default creation of relieved bends in a model (see Section 2.4.4.4.5 on p. 271).
Rectangular, Rounded	Select the default option for the shape of release.
Depth, Width	Enter the default dimensions for release of bend.
Flat pattern	This group of options allows to specify how lengths of flat pattern of bends should be distributed.
The coefficient that determines position of the neutral layer	Select this option if the length of a flat pattern should be calculated using the coefficient of position of the neutral layer (see Section 2.4.1.4.1 on p. 237). Enter the default value of the coefficient.
Bend amount	Select this option if the length of a flat pattern will be specified explicitly (see Section 2.4.1.4.2 on p. 238) and enter the default value of the bend amount.
Reduction of bend	Select this option if the length of a flat pattern should be defined by specifying the bend reduction (see Section 2.4.1.4.3 on p. 238). Enter the default value of the bend reduction.

Tab. 9.2.59. Control elements of the dialog to setup parameters of a sheet body

Element	Setting description
	<p>Bend Table</p> <p>Enable this option if the lengths of flat pattern of bends should be taken from the bend table (see Section 2.4.1.4.4 on p. 239). To choose a file from the table, press the “...” button and specify the desired file in the standard dialog of choosing a file. The full name of the specified file will appear in the Bend table field.</p> <p>Note that when the Bend table option is enabled, the options for determining the length of a flat pattern become inaccessible, because the use of the table have higher priority than those options. At the same time, the numerical fields in the Flat pattern group remain accessible. The values entered into them will be applied by default for bends that use the respective variants of determining the length of a flat pattern.</p>

- * While building a sheet body, bends that correspond to contour angles in the sketch are built according to inner radius, irrespective of the status of this option.

Setting of parameters of sheet bodies for the current model has the following specific features. All numerical fields in the properties dialog of the current sheet body (**Width**, **Bend radius**, etc.) are for reference. Those correspond to the current values of the variables of the sheet body (see Section 2.4.1.3 on p. 233). When the value of a variable is changed, the contents of the respective field is changed accordingly.

Thus, the following settings are possible directly in the sheet body properties dialog of the current model:

- ▼ changing the method through which bend radii are set,
- ▼ changing the angle interpretation,
- ▼ enabling or disabling bend releases and selecting the shape of release,
- ▼ selecting the method through which length of flat pattern are determined,
- ▼ changing the bend table.

After parameters of a sheet body are set, click the **OK** button. To leave the dialog without making changes to the settings, click the **Cancel** button.

The newly set parameters will be used as default parameters when creating new bends in a sheet body. Parameters of bends that were created before the changes to the settings will remain unchanged.



Selection of a new table of bends will affect the bends that already exist in a model. You can find more information on this in Section *Change of the Table of Bends* on p. 240.

9.2.7.4.9. General tolerances

Setup of general tolerances for parts' dimensions (both new and current ones) is performed in the **General tolerances (Unspecified maximum deviations of dimension)**. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — General tolerances,**
- ▼ for the current document — **Service — Parameters... — Current part — General tolerances.**

A similar dialog is used when setting up general tolerances for dimensions of simple and technological assemblies.

Tab. 9.2.60. Controls of the general tolerance setup dialog

Element	Setting description
Use general tolerances under GOST 30893.1 (ISO2768-1)	If the option is enabled, general tolerances will be assigned to dimensions of sketches and model operations with unspecified individual tolerances. If the option is disabled, no general tolerance will be assigned; dimension will either have no tolerance or an individual tolerance. More on dimensions of sketches and operations, see Section 2.9 on p. 631; on tolerances, Section 2.10 on p. 667.
Accuracy class	You can use this list to choose the accuracy class that defines values of general tolerances.

After the setting is finished, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.10. Accuracy of displaying and mass-centering properties (MPs)

Parameters for accuracy of displaying and calculation of mass-centering properties (MPs) as well as accuracy of calculation of faces in parts (both new and current) are set up in the **Accuracy of displaying and MPs** dialog. The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — Accuracy of displaying and MPs command,**
- ▼ for the current document — **Service — Parameters... — Current Model — Drawing Precision and MPs command.**

A similar dialog is used to set the accuracy of displaying and calculation of mass-centering characteristics and areas of faces in simple and technological assemblies.

Tab. 9.2.61. Control elements of the dialog to setup the accuracy of displaying and calculation of MPs

Element	Setting description
Accuracy of displaying	<p>Serves to increase or decrease the accuracy of approximating curvilinear edges and model silhouette lines by a straight line segments, and curved surfaces with triangles.</p> <p>A change in the accuracy of drawing of a model affects only displaying of bodies created in this model and inserted into it as templates and surfaces created in this model or imported into it. The precision of model component drawing should be set up in component files.</p> <p>To change the accuracy, move the slider across the range of positions from Rough to Fine. The information field of the dialog will display a conventional parameter of accuracy of displaying— the number of triangles, the preview window will display the sphere surface with the specified accuracy.</p> <p>The higher the accuracy, the “smoother” is the appearance of the image and the more time is required for calculations.</p>
Accuracy of calculations of MPs	<p>Allows to increase or decrease the accuracy of calculating mass-centering properties (MPs) of the model and 3D faces area.</p> <p>To change the accuracy, move the slider between the Low and High positions.</p> <p>The higher the accuracy, the more time it takes to run calculations. We do not recommend to use the maximum accuracy without a paramount necessity, because calculations with this accuracy takes a considerable time.</p> <p>Calculations of areas in the current model (part or assembly) are run with the accuracy set for this model.</p> <p>MPs in a model containing the components are calculated with a precision specified for the model. The precision set for components during their creation or editing is ignored.</p>

After the setting is finished, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.11. Model tree — General display settings

Displaying of the Design Tree for parts (both new and current) is set up in the **General display settings**. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Model tree — General display settings**,
- ▼ for the current document — **Service — Parameters... — Current part — Model tree — General display settings command**.

The same dialog is used when adjusting of the Design Tree for simple and technological assemblies.

The dialog is used to select the saved view of the Design Tree and enable or disable a grouping of components.

Tab. 9.2.62. Control elements of the dialog for the Design Tree's display general settings

Element	Setting description
Tree appearance to be saved	<p>The group of options allows to choose the way of displaying objects in the Design Tree — Displaying of structure (see Section 2.1.2.2 on p. 106) or Displaying of sequence of construction (see Section 2.1.2.3 on p. 108).</p> <p>When customizing the current model, the chosen way determines the savable Tree appearance, i.e. the way which will be saved when the file is closed and will be used when this model will be opened.</p> <p>When customizing new models, the chosen way of displaying will be used as default.</p>
?Group components	<p>This option is used to divide the Design Tree's components into groups by type — <i>Assembly units, Parts, Library components, Local parts, Standard items</i>.</p> <p>“Empty” groups are not displayed. The group appears in the Tree after creation of the first object of the respective type.</p> <p>When you customize the current model, changes will be displayed in the Tree:</p> <p>When customizing new models, the chosen variant will be used as default.</p>



When you first open a model created in KOMPAS-3D V8 Plus or an earlier system version, the Tree is displayed as it was set during customization of new models. The same way of displaying is taken as the savable Tree appearance. You may change the savable Tree appearance, if necessary.

After a model is saved in KOMPAS-3D V9 and later versions, the information about the savable Tree appearance will be written to the file of this model and will be used when the model will be opened.

If a model is not saved, the Tree will be again displayed in the way as set for new models when this model is opened.

After the Tree is customized, click the **OK** button.

To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.12. Model Tree — Structure

Displaying of the Design Tree structure for parts (both new and current) is set up in the **Displaying in the Tree**. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Model tree — Structure**,

- ▼ for the current document — **Service — Parameters... — Current part — Model Tree — Structure** command or by clicking the **Model Tree Composition** button on the Design Tree's toolbar.



The same dialog is used when adjusting of the Design Tree structure for simple and technological assemblies.

The dialog makes it possible to set up the default composition of the Design Tree.

Tab. 9.2.63. Control elements of the dialog to setup the structure of the Design Tree

Element	Setting description
Displaying of structure	A group of options which correspond to Model Tree sections. You can enable and disable displaying of sections in the Design Tree by enabling and displaying corresponding options. “Empty” sections are not displayed. The section appears in the Design Tree after creation of the first object of the respective type. For instance, if a model does not have a single sketch, there will be no Sketches section in the Tree.



When you first open a model created in KOMPAS-3D V8 Plus or an earlier system version, the Tree is displayed as it was set during customization of new models.

After the Tree is customized, click the **OK** button. When you customize the current model, changes will be displayed in the Tree: the respective sections will disappear or appear in it. When customizing new models, changes will take effect during creation of these models.

To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.13. Model Tree — Name display format

Displaying of the Design Tree's name for parts (both new and current) is set up in the **Name display format** dialog. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — Model tree — Name display format**,
- ▼ for the current document — **Service — Parameters... — Current part — Model tree — Name display format command**.

The same dialog is used for setting up displaying root object name and components in the Design Tree for simple or technological assemblies.

Tab. 9.2.64. Control elements of the name display format setup dialog

Element	Setting description
	Object names in the Tree may contain one or two parts (designation and name) of text, which is entered in the Designation and Name fields on the Property bar when setting up object properties (see Section 5.1.4.1.1 on p. 1385).

Tab. 9.2.64. Control elements of the name display format setup dialog

Element	Setting description
Name	Choose this variant for objects' names to be used as names. If a name of an object is not specified, the name of the object's type will be used.
Designation + Name	Choose one of these variants for object's names and designations, separated by spaces, to be used as objects' names.
Name + Designation,	If either name part is not specified, the other part is used without a space. If neither name part is specified, the name of the object's type is offered as the object's name.

After the Tree is customized, click the **OK** button.

To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.4.14. Zones

The list of objects defining zone sizes is set up for the current parts (assemblies) in the **Zones by object sizes** dialog called by **Service — Parameters... — Current part/assembly — Zones**.

The dialog is used to set objects by which zone sizes are determined when a composite object, for example, a component is indicated for zone creation.



The setup performed in this dialog does not affect the manual selection of objects defining zone size, i.e. an object of any type can be selected manually.

Tab. 9.2.65. Controls of the dialog of zone setup by object sizes

Element	Setting description
Elements used in defining the size	The list of objects used in defining zone size. Enable the option to the left of the object name for it to be included in zone size definition.
Respect hidden objects	Enable the option to include hidden objects in zone size definition.

To finish the setup, click the **OK** button. To leave the dialog without saving change, click the **Cancel** button.

9.2.7.4.15. Units for MPs setting

Measurement units for MPs for parts (both new and current) are set up in the **Units for MPs setting** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Part — Units for MPs setting command**,

- ▼ for the current document — **Service — Parameters... — Current Model — Units of MPs setting** command.

The same dialog is used when adjusting of the measurement units of MP for simple and technological assemblies.

The dialog allows to set the default measurement units of length and mass for entering and displaying the mass-centering properties of a models while setting its properties.



When calculating MPs using the special command (see Section 2.1.2.2 on p. 106), the measurement units are set in the process of execution thereof and may differ from the units set in the above dialog.

Choose the desired names from the **Length units** and **Mass units** lists to set the units.

After setup is completed, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.5. Assembly

9.2.7.5.1. Properties

New assemblies' properties are set in the **Assembly's properties** dialog which is called by the **Service — Parameters... — New Documents — Model — Assembly— Properties command**,

A similar dialog is used when setting up properties for new technological assemblies.

Tab. 9.2.66. Control elements of the Properties Setup dialog

Element	Setting description
Designation	The field allows to enter the designation which will be assigned to an assembly by default.
Name	The field allows to enter the designation which will be assigned to an assembly by default.

When properties of an assembly are set up, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.2.7.5.2. Customization of properties list

You can set up connection to necessary libraries of properties and displaying of properties (both new and current) in assemblies. To do this, use the **Customization of properties list** dialog. This dialog is displayed in the following cases:

- ▼ for new documents — after calling the **Service — Parameters... — New Documents — Model — Property list setup** command,
- ▼ for the current document — after calling the **Service — Parameters... — Current assembly — Customization of properties list** command, as well as after pressing the button **Custom-**



ization of property list on the Property panel when properties of an assembly, component, or body are set.

For new documents, connection to necessary property libraries and displaying of properties in a document are customized in the dialog. For current document, only displaying of properties in a document is customized.

Working with the dialog is described in Section 9.2.6.5 on p. 1762.

9.2.7.5.3. New versions' numbers

Parameters of forming the numbers of new versions are set up in the **New versions' numbers** dialog. This dialog is displayed in the following cases:

- ▼ for new documents — after calling the **Service — Parameters... — New Documents — Model — Assembly — New version's numbers** command,
- ▼ for the current document — after calling the **Service — Parameters... — Current Assembly — New versions' numbers** command, as well as after clicking the **Parameters of forming new versions' numbers** button in the Document Manager when working with versions and additional numbers.



Working with the dialog is described in Section 9.2.7.4.3 on p. 1845.

9.2.7.5.4. Color

Color and optical properties of new assemblies' surfaces are set in the **Assembly's color** dialog which is called by the **Service — Parameters... — New Documents — Model — Assembly — Color** command,

Note that parts in an assembly preserve their own colors. Settings made in the dialog only relate to operations of adding or removal of material that are carried out in an assembly itself.

Work with dialog is described in Section 9.2.7.4.4 on p. 1847.

9.2.7.5.5. Properties of the absolute coordinate systems

Properties of the absolute coordinate system for assemblies (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Assembly — Properties of absolute CS** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Properties of absolute CS**.

Working with the dialog is described in Section 9.2.7.4.5 on p. 1848.

9.2.7.5.6. Properties of local coordinate systems

Properties of local coordinate systems for assemblies (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Assembly — Properties of local CS** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Properties of local CS**.

Working with the dialog is described in Section 9.2.7.4.6 on p. 1849.

9.2.7.5.7. Object properties

Object properties for assemblies (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Assembly — Properties of objects** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Properties of objects**.

Working with the dialog is described in Section 9.2.7.4.7 on p. 1850.

9.2.7.5.8. General tolerances

Setup of general tolerances for assemblies' dimensions (both new and current ones) is performed in the **General tolerances (Unspecified maximum deviations of dimension)**. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Assembly— General tolerances**,
- ▼ for the current document — **Service — Parameters... — Current part — Model — Assembly — General tolerances**,

Working with the dialog is described in Section 9.2.7.4.9 on p. 1854.

9.2.7.5.9. Accuracy of displaying and MPs

Parameters for accuracy of displaying and calculation of mass-centering properties (MPs) as well as accuracy of calculation of faces in assemblies (both new and current) are set up in the **Accuracy of displaying and MPs** dialog. The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Assembly — Accuracy of displaying and MPs** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Drawing Precision and MPs** command.

Working with the dialog is described in Section 9.2.7.4.10 on p. 1854.

9.2.7.5.10. Model Tree

Displaying of the Design Tree for assemblies (both new and current) is set up in the **Displaying in the Tree** dialog. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Assembly — Model tree** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Model Tree** command or by clicking the **Design Tree Composition** button on the Model Tree's toolbar.



Working with the dialog is described in Section 9.2.7.4.12 on p. 1856.

9.2.7.5.11. Zones

The list of objects defining zone sizes is set up for the current assemblies in the **Zones by object sizes** dialog called by **Service — Parameters... — Current assembly — Zones**. Working with the dialog is described in Section 9.2.7.4.14 on p. 1858.

9.2.7.5.12. Units for MPs setting

Measurement units for MPs for assemblies (both new and current) are set up in the **Units for MPs setting** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Assembly — Units for MPs setting command**,
- ▼ for the current document — **Service — Parameters... — Current assembly — Units of MPs setting command**.

Working with the dialog is described in Section 9.2.7.4.15 on p. 1858.

9.2.7.6. Technological assembly

9.2.7.6.1. Properties

New technological assemblies' properties are set in the **Assembly's properties** dialog which is called by the **Service — Parameters... — New Documents — Model — Technological Assembly— Properties command**,

Working with the dialog is described in Section 9.2.7.5.1 on p. 1859.

9.2.7.6.2. Customization of properties list

You can set up connection to necessary libraries of properties and displaying of properties (both new and current) in technological assemblies. To do this, use the **Customization of properties list** dialog. This dialog is displayed in the following cases:

- ▼ for new documents — after calling the **Service — Parameters... — New Documents — Model — Technological assembly — Property list setup command**,
- ▼ for the current document — after calling the **Service — Parameters... — Current assembly — Customization of properties list** command, as well as after pressing the button **Customization of property list** on the Property panel when properties of an assembly, component, or body are set.



For new documents, connection to necessary property libraries and displaying of properties in a document are customized in the dialog. For current document, only displaying of properties in a document is customized.

Working with the dialog is described in Section 9.2.6.5 on p. 1762.

9.2.7.6.3. New versions' numbers

Parameters of generating new versions' numbers are set in the dialog called by the **Service — Parameters... command**. — **New Documents — Model — Technological Assembly — New version's numbers** command,

Working with the dialog is described in Section 9.2.7.4.3 on p. 1845.

9.2.7.6.4. Color

Color and optical properties of new technological assemblies' surfaces are set in the **Assembly's color** dialog which is called by the **Service — Parameters... — New Documents — Model — Technological Assembly— Color** command,

Note that parts in a technological assembly retain their own colors. Settings made in the dialog only relate to operations of adding or removal of material that are carried out in a technological assembly itself.

Work with dialog is described in Section 9.2.7.4.4 on p. 1847.

9.2.7.6.5. Properties of the absolute coordinate system

Properties of the absolute coordinate system for technological assemblies (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Technological Assembly — Properties of absolute CS** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Properties of absolute CS**.

Working with the dialog is described in Section 9.2.7.4.5 on p. 1848.

9.2.7.6.6. Local CS properties

Properties of local coordinate systems for technological assemblies (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Technological Assembly — Properties of local CS** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Properties of local CS**.

Working with the dialog is described in Section 9.2.7.4.6 on p. 1849.

9.2.7.6.7. Object properties

Object properties for technological assemblies (both new and current) are set up in the **Object properties** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Technological Assembly— Object Properties** command,
- ▼ for the current document — **Service — Parameters... — Current assembly — Properties of objects**.

Working with the dialog is described in Section 9.2.7.4.7 on p. 1850.

9.2.7.6.8. General tolerances

Setup of general tolerances for technological assemblies' dimensions (both new and current ones) is performed in the **General tolerances (Unspecified maximum deviations of dimension)**. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Technological Assembly— General tolerances**,

- ▼ for the current document — **Service — Parameters... — Current part — Model — Assembly — General tolerances,**

Working with the dialog is described in Section 9.2.7.4.9 on p. 1854.

9.2.7.6.9. Accuracy of MP drawing

Parameters for accuracy of displaying and calculation of mass-centering properties (MPs) as well as accuracy of calculation of faces in technological assemblies (both new and current) are set up in the **Accuracy of displaying and MPs** dialog. The dialog is invoked by the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Technological Assembly — Accuracy of displaying and MPs command,**
- ▼ for the current document — **Service — Parameters... — Current assembly — Drawing Precision and MPs command.**

Working with the dialog is described in Section 9.2.7.4.10 on p. 1854.

9.2.7.6.10. Model Tree

Displaying of the Design Tree for technological assemblies (both new and current) is set up in the **Displaying in the Tree** dialog. The dialog is called using the following methods:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Technological Assembly — Model tree command,**
- ▼ for the current document — **Service — Parameters... — Current assembly — Model Tree command** or by clicking the **Design Tree Composition** button on the Model Tree's toolbar.



Working with the dialog is described in Section 9.2.7.4.12 on p. 1856.

9.2.7.6.11. Units for MPs setting

Measurement units for MPs for technological assemblies (both new and current) are set up in the **Units for MPs setting** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Technological Assembly — Units for MPs setting command,**
- ▼ for the current document — **Service — Parameters... — Current assembly — Units of MPs setting command.**

Working with the dialog is described in Section 9.2.7.4.15 on p. 1858.

9.2.7.7. Sketch

9.2.7.7.1. Parametrization

The use of the parametric mode of the KOMPAS-3D system for sketches of models (both new and current) is customized in the **Parametrization management** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New Documents — Model — Sketch — Parametrization command,**

- ▼ for the current document — **Service — Parameters... — Current sketch — Parametrization.**

Working with the dialog is described in Section 9.2.6.21 on p. 1830.

9.2.7.8. Report table parameters

9.2.7.8.1. Title

Parameters of text for table title in a report for models (both new and current) are set up in the **Parameters of text for table title** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Model — Report table parameters — Heading,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Report table parameters — Heading.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

The settings are used when displaying a report on a model in the Data Processing Window.

9.2.7.8.2. Cell

Parameters of cell text for table in a report for models (both new and current) are set up in the **Parameters of text for table cell** dialog invoked with the following command:

- ▼ for new documents — **Service — Parameters... — New documents — Model — Report table parameters — Cell,**
- ▼ for the current document — **Service — Parameters... — Current part/assembly — Report table parameters — Cell.**

Work with dialog is described in Section 9.2.4.3 on p. 1752.

The settings are used when displaying a report on a model in the Data Processing Window.

9.2.7.9. Technical specifications

Parameters of the technical specifications' text that are set in the model, in case of new document, are adjusted in the dialog called by the **Service — Parameters... command — New Documents — Model — Technical specifications** command,

Tab. 9.2.67. Control elements of the technical specifications' text parameters setup dialog

Element	Setting description
Line spacing, mm	The field to enter the distance between lines in paragraph. Changing the line spacing makes it possible to retain proportional spaces between lines that have fonts with different symbol height.
First line Indent, mm	The field to enter paragraph first line offset. Indent is measured from text boundary. When creating a new paragraph, the cursor is automatically set to a position with preset indent.

Tab. 9.2.67. Control elements of the technical specifications' text parameters setup dialog

Element	Setting description
Indentations	The fields for entering distances between left and right input field boundaries and corresponding text borders.
Spacing	The field to enter the distance between two contiguous paragraphs. The given distance is formed by two values— the interval after the first paragraph and the interval before the second paragraph.
Alignment	Options serve to define desired paragraph alignment method (left, right, centered, justified).
Set Numeration Upon Input	The option controls the auto-numbering of the specifications paragraphs on entry. If checked, the paragraphs are numbered automatically.
Font...	A button to call the font parameter setup dialog.
Tabs...	A button to call the dialog to set up tabulation parameters.

Upon completion of setting up parameters, click the **OK** button. To leave the dialog without saving changes, click the **Cancel** button.

9.3. Current window parameters

9.3.1. Grid

The grid view displayed in the active window can be configured in the **Grid Setup** dialog that opens when you call the command **Service — Options... — Current window— Grid** or **Set up Parameters** from the **Grid** button menu, located on the **Current Status** bar. Working with the dialog is described in Section 9.1.7.2 on p. 1705.

9.3.2. Scroll Bars

Controlling the scroll bar display and the image auto shift in the current window is done in the **Scroll Bar Setup** dialog called with the **Service— Parameters— Current Window— Scroll Bars** command. Working with the dialog is described in Section 9.1.7.3 on p. 1706.

9.3.3. Perspective Projection

You can setup perspective projection parameters (degree of distortion introduced by the perspective) for the current window in the **Perspective Projection** dialog, called with the **Service— Parameters— Current Window— Perspective Projection**.

Dialog contains **Distance in Model Size** field. The value in this field shows, by how many times the distance from the model to the image plane is greater than the maximum model dimension. In other words, the model is shown on the screen in such a way that could be obtained by an optical instrument located at the specified distance from the model. The lesser the specified distance is, the more noticeable will be the image distortion.

To finish the setup, click the **OK** button. To leave the dialog without saving change, click the **Cancel** button.

9.3.4. Zones

Zone display parameters for the current window are set up in the **Zone display** dialog called by the **Service — Parameters... — Current window — Zones**.

The controls of this dialog are described in Table 9.3.1.

Tab. 9.3.1. Controls of the zone display setup dialog

Element	Setting description
Edge display	This option is used to manage the display of edges of a polyhedron that circumscribes the zone sizes.

Tab. 9.3.1. Controls of the zone display setup dialog

Element	Setting description
Color	The button allows to choose the color for the zone polyhedron edges in the standard color selection dialog. Available if the Edge display option is enabled.
Line	The list for line style selection showing edges of the zone polyhedron. Edges can be shown with: <ul style="list-style-type: none"> ▼ a solid line, ▼ a dashed line, ▼ a dotted line. The list is available if the Edge display option is enabled.
Paint faces	This option is used to manage the display of faces of a polyhedron that circumscribes the zone sizes.
Color	The button is used to select the color of the zone polyhedron in the standard color selection dialog. Available when the Paint faces option is enabled.
Optical Properties	A set of controls allows to setup optical properties of the surface of zone polyhedron faces. Available if the Paint faces option is enabled.
Ambient	To define values of the parameters, position each slider as appropriate.
Diffusion	The numerical value of a parameter is displayed in the information field.
Specularity	
Shininess	
Transparency	
Emission	
Preview Window	The preview window displays a sphere with the color and properties of zone polyhedron faces specified in the dialog. This helps to make visual evaluation of introduced changes.

To finish the setup, click the **OK** button. To leave the dialog without saving change, click the **Cancel** button.

9.4. Profiles. Interface setup. Application type

9.4.1. Profiles

A **profile** is a collection of data about system configuration settings.

Using profiles you can quickly modify the current system configuration.

For example, you may create a profile for 3D simulation (i.e. a profile containing settings to work more conveniently with 3D simulations), a profile to develop design documentation, etc. Then, to set up the system for performing a particular task, you will just need to apply the appropriate profile.

In order to save the current configuration setup in the profile, do the following.

1. Call the **Service — Profiles...** command.
2. In the appearing dialog, enable the options that correspond to the groups of settings that should be saved in the profile.
3. Click the **Save as...** button.
4. In the appearing dialog, specify the name and location of the profile file. A profile file has an extension of *pfl*.



The profile file does not record the names of the last opened documents, nor does it record the configuration of the open document windows.

In order to apply previously created profile, do the following.

1. Call the **Service — Profiles...** command.
2. In the dialog that appears, select the profile from the **Profile** list.
3. Enable the options corresponding to setup groups that should be used.
4. Click the **Apply** button.
5. The specified settings will be applied.
6. Click the **Exit** button.

Several predefined profiles are supplied together with KOMPAS-3D. Those are located in the *Profiles* subfolder of the main system folder.

When using ready-made profile files, keep in mind that they do not contain the references to libraries. Therefore, after any such profile is applied, the current contents of the Library Manager will be deleted, and the Manager will automatically fill according to the **.lms* files located in the *Sys* subfolder of the system root folder.

The **.lms* files contain information about the contents of Library Manager (names of sections, subsections, and lists of libraries they include). These files are used for the first completion of the Library Manager and its updating. Text format is used in the files, therefore, you can open them in any text editor and modify the available data.

9.4.2. Customizing interface

Fig. 1.2.1 on p. 51 shows the default KOMPAS-3D screen view. You can set up the system interface at your own discretion. To do so, use the **Service — Interface setup** command. A set-up dialog with the open **Screen — Interface setup** section will appear on the screen. The section contains the following items:

- ▼ **Commands,**
- ▼ **Toolbars,**
- ▼ **Utilities,**
- ▼ **Keyboard,**
- ▼ **Menus,**
- ▼ **Parameters,**
- ▼ **Icon size.**

Upon selection of each of the items, a dialog with the respective section name appears in the right part of the tab. You can use the interface items located in the dialogs to make the required settings.

Henceforth, this section's items are referred to when describing the interface element setup.

The KOMPAS-3D interface setup procedure is in many ways similar to setup of other Windows applications; therefore, we don't discuss each option in detail.



Note that it is impossible to cancel a modified setting (except for changing icon size). To return the interface to the previous view, the corresponding setting shall be changed again.

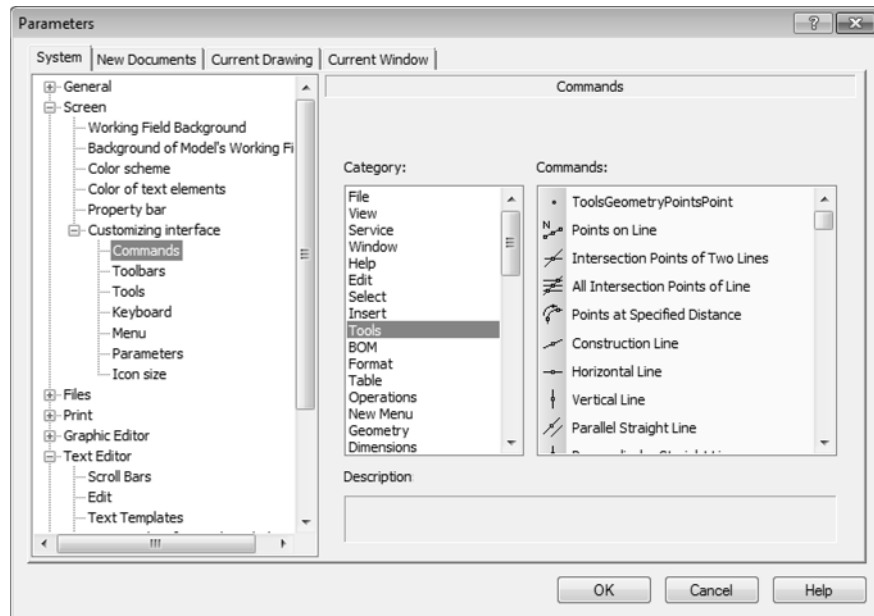
You can additionally adjust displaying of commands in menus and buttons on toolbars using context menu.

In order to work with the context menu, open the **Parameters** dialog, choose one of the **Interface setup** section items and, without closing the dialog, call the context menu of the element that needs to be set up.

Context menu commands make it possible to choose the element display option — in the form of an icon, text, icon and text, replace icon by a custom one; copy icon and so on.

9.4.2.1. Setup of the Main menu and toolbar components

Select the **Commands** item (Fig. 9.4.1). In the **Commands** dialog, select the category and command name. “Drag” command to the desired bar or menu.

Fig. 9.4.1. **Commands** dialog

You can change commands' and their groups' positions in toolbars and in the menu by dragging them with the mouse. The command's (button's) context menu can be used for additional setup (changing the command's name, its icon, etc.) and also for deletion of the selected element.

The **Reset all** button in the **Toolbar** dialog makes it possible to restore the default settings of all menus and toolbars.



You can also move buttons between bars without calling the setup dialog. To do this, drag them by the mouse keeping the <Alt> key pressed.



It is impossible to change settings of toolbars in dialogs and windows.

9.4.2.2. Creation of a custom toolbar

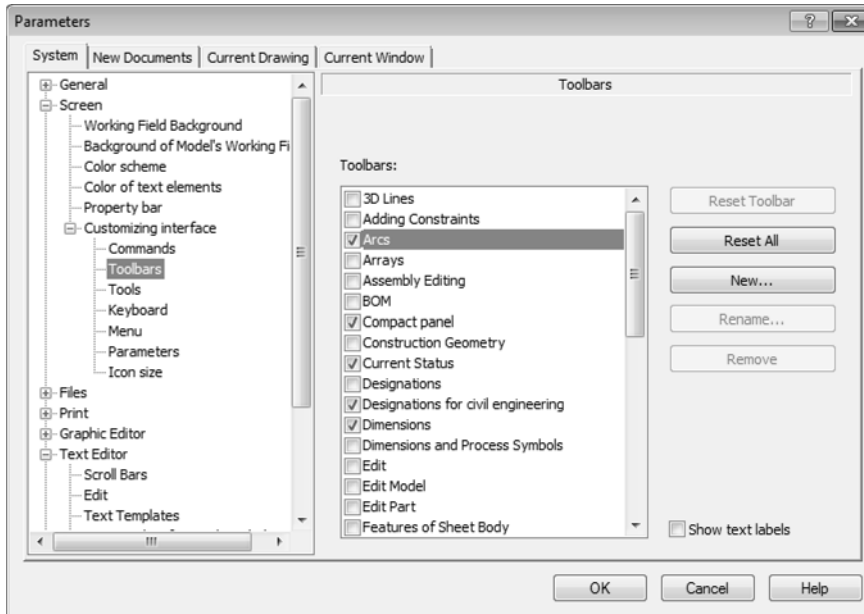


Fig. 9.4.2. **Toolbar** Command

You can create user-defined toolbars and place on them any commands.

In order to create a custom toolbar, choose the **Toolbars** item and click **New...** (Fig. 9.4.2). Enter the new toolbar's name. The new toolbar will appear on the screen.

The toolbar will be empty at first. Set up the toolbar's elements as is described in Section 9.4.2.1.

If you need to insert a separator between commands, highlight the command in front of which the separator should be positioned on the bar and call the **Begin group** command from the context menu. The separator will appear in front of the chosen command.

9.4.2.3. Keyboard Shortcut Settings ("hot keys")

Choose the **Keyboard** item.

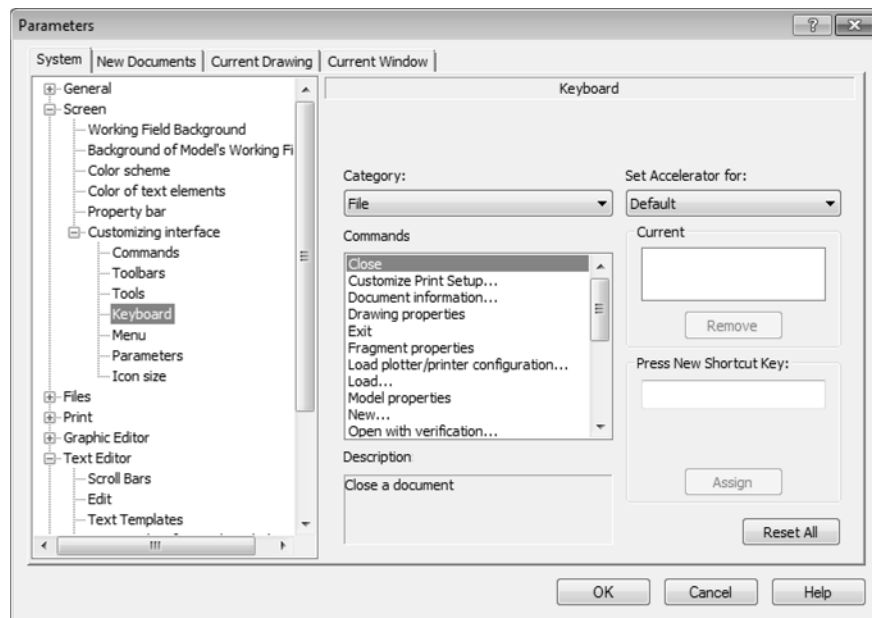


Fig. 9.4.3. **Keyboard** dialog

Select a category and a command from this category (Fig. 9.4.3). If a key combination is already assigned to the command, it will be displayed in the **Current** list. Several key combinations may be assigned to one command.

From the **Assign combination** list, choose the mode in which the combination will be active.

Assign a **new key combination**. You can use both single keys, for example, letters and numbers, and their combinations with **<Ctrl>**, **<Alt>** keys or with key combinations **<Ctrl> + <Alt>**, **<Ctrl> + <Shift>**, **<Shift> + <Alt>**. Combinations of keys help avoid confusion when calling commands or entering letters and numbers. It is impossible to assign a key or key combination that is already used for calling a command.

Confirm the new combination assigned by clicking **Assign**. In order to cancel using a key combination, highlight it in the **Current** list and click the **Delete** button.

In order to restore the default values of key combinations for all commands, click **Reset all**.



Note that numerical keypad keys operate in different ways depending on whether the NumLoc mode is enabled or disabled. If binding keypad keys with commands was made with the NumLoc mode enabled (disabled), the NumLoc mode should also be enabled (disabled) for calling the command by clicking the respective key.

9.4.2.4. Utilities

In the KOMPAS-3D **Service** menu, you can add custom commands for opening application files, documents, etc. (Fig. 9.4.4).

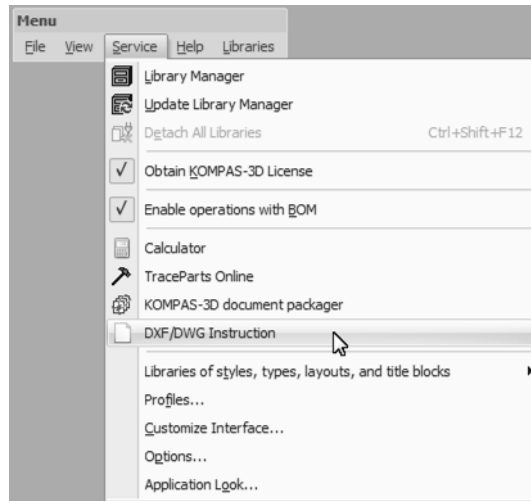


Fig. 9.4.4. Running utilities from the **Service** menu

In order to add a new command to the menu, choose the **Utilities** item (Fig. 9.4.5).

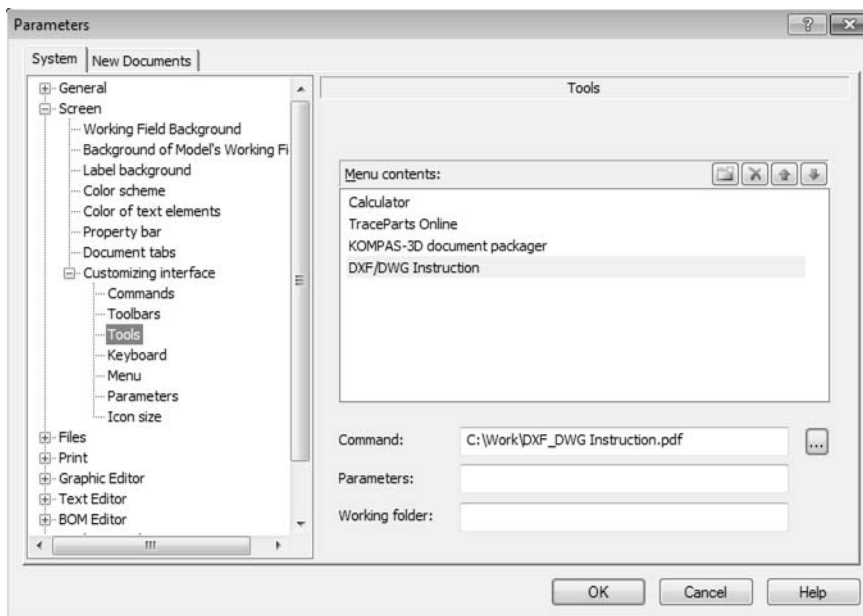


Fig. 9.4.5. **Utilities** dialog

Click the **New** button.

Enter the command name in the empty line that appears in the list.

In the **Command** field: enter the full name of the file to be opened or run when the command is called. In order to avoid typos, use the **Browse** button to select the required file.

If an executable file is selected, then the created command shall launch the application. If a non-executable file is selected, then the created command shall launch the Windows application associated to that file type. The file will be opened in this application.

In the **Parameters** field, enter application launch parameters.

In the **Working folder** field, enter the path to the folder of the application which is started by the command.



If, after calling the command from the menu (Fig. 9.4.4), the file does not run, make sure the path to the file is correct. The path can be invalid if, for example, you use a profile, created at some other workstation.

9.4.2.5. Icon size setup

Menu commands and command calling buttons icon size can be changed in the toolbars. To do this, use the **Icon size** item.

Choose icon sizes for each type from the lists.



Menu commands and toolbar buttons can be displayed as icons, text or icons with text. Selection of display option is described in Section 9.4.2 on p. 1872.

9.4.3. Application display

You can choose the display style for the system window elements in the special dialog (Fig. 9.4.6).

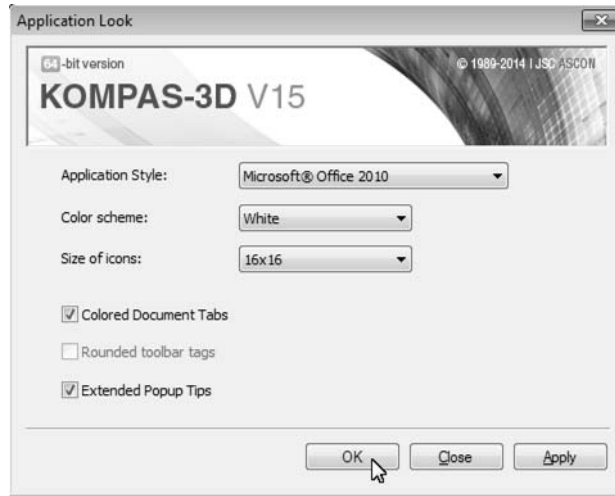


Fig. 9.4.6. Select Application View dialog

This dialog is displayed on the first launch of the KOMPAS-3D system or after calling the **Service — Application Look...** command.

The control elements of this dialog are represented in table 9.4.1.



Upon the launch of the system, the application appearance selection dialog is displayed without the **Close** button.

Tab. 9.4.1. Control elements for application style selection dialog

Element	Description
Application Style	The list from which you can select an application style. Select one of the variants from the drop-down list.
Color Scheme	The list from which you can select an application color scheme. The list is available if the application style is set to Microsoft® Office 2007 or Microsoft® Office 2010 . Select one of the variants from the drop-down list.
Icon size	List to select the size of icons on panels, in menu and in the Document Tree. If in Icon size setting dialog (see Section 9.1.3.8.7 on p. 1690) option Single is disabled, then in the Icon size list the According to settings row appears. If it is selected, icons will have the size set in the icon size setup dialog.

Tab. 9.4.1. Control elements for application style selection dialog

Element	Description
&Colored document tabs	The option controls the style of document tabs. When activating this option, the tabs are displayed in different colors, and the tabs scroll button is located at the left of them.
Rounded Toolbar Tabs	The option controlling toolbar tag style. This option is available if the application style is set to Microsoft® Visual Studio 2005 . If this option is enabled, toolbar tags are displayed as rounded when the toolbars are minimized. On minimizing bars and windows, see Section 1.2.2.3.3 on p. 58.
Extended Pop-up Tips	The option controlling look the appearance of pop-up tips. If the option is enabled, information from the message line is added to the pop-up tip that appears when the mouse point is over the working window element. If the option is disabled, the pop-up tip will contain a command name only.
Show this Dialog at Startup	The option controls automatic opening of the dialog. If it is enabled, then the dialog appears each time after launching the system.
OK	Clicking this button closes the dialog and applies the selected settings.
Close	Clicking this button closes the dialog without applying the selected settings. The same result is achieved by closing the dialog by the button on its title.
Apply	Clicking this button applies the selected settings without closing the dialog.

9.5. Storing of system settings

9.5.1. General information

When the KOMPAS-3D is running, the so-called *system files and folders* are used as well as the files of the system itself (located in the \Bin subfolder of the main system folder). These include:

- ▼ configuration files, containing information on system parameters, system windows, and new documents setup (see Section 9.5.2 on p. 1881),
- ▼ default folders to place certain type files (libraries of styles and layouts, templates, profiles etc.).

Path to the system files and folders is set up upon launching KOMPAS-3D and is stored in the form of **KOMPAS-3D environment variables** till the end of session. The procedure to define paths to system files and folders is described in Section 9.5.4 on p. 1886.

If necessary, you can specify paths to system files and folders in *KOMPAS.ini* (see Section 9.5.3 on p. 1883).

At any time while working with the system, you can get information about current location of *KOMPASini* file, system files and folders in the file location view dialog (see Section 9.1.4.1 on p. 1691). This dialog also allows to create or edit *KOMPAS.ini* file.



A new newly created or edited *KOMPAS.ini* file will be used in the next KOMPAS-3D working session.

Information not related to the system settings but required for KOMPAS work are stored in the Registry. It is the following information:

- ▼ user data,
- ▼ list of the most recently opened files,
- ▼ current screen resolution (you need this data to snap main system window).



Current document parameter settings (settings performed on the **Current Document** tab of the general dialog for parameter setting up) are stored in the document itself. Due to this option, they will not change when transferring documents from one working station to another.

9.5.2. Configuration files

Configuration files contain data about KOMPAS-3D system configuration settings. Configuration setting up includes three setting groups, so there are also three configuration files: **.cfg*, **.prj* and **.dsk* (see Table 9.5.1).

Tab. 9.5.1. KOMPAS-3D configuration files and settings stored in them

Configuration file	Settings stored in a file
.cfg	<p>System settings Settings made on System tab of the general setup dialog [] Interface settings are an exception (they are performed in Section Screen — Customizing interface). These settings are saved in <i>*.dsk</i> file. Section Files— Location are designed to view, so it is not included into <i>*.cfg</i> file. See a detailed description of system settings in Section 9.1 on p. 1667.</p>
.prj	<p>New document settings Settings made on New Documents tab of the general setup dialog [] See a detailed description of new document settings in Section 9.2 on p. 1745.</p>
.dsk	<p>Active Window Settings:</p> <ul style="list-style-type: none"> ▼ Main Window Size ▼ Interface settings made in the Screen — Interface Settings section of the System tab in the general settings dialog [], ▼ Application look ▼ status of open documents (window size and location, cursor step, scale, local snaps, grid and scroll bar settings), ▼ location and composition of the menu bar and toolbars (including the custom ones) ^{**}, ▼ position and size: <ul style="list-style-type: none"> ▼ Properties Bar ▼ variables windows, ▼ Information window, ▼ Text Template Library window ▼ Library windows ▼ Document Manager and Layer Filter Settings dialog, ▼ Library Manager ▼ Library Manager structure ▼ Each library status: connected or disconnected.

* The dialog invoked by the **Service— Parameters...** command.

** Information about contents of **View** and **Current Status** toolbars is not stored in the editing local fragment, editing technical specifications, editing BOM items in graphic document modes. Changes to these toolbars in these modes are stored only until the end of session.

There are no configuration files right after system installation. They are created after the first change to settings is made and saved. Files for configuration files are defined as is described in Section 9.5.4 on p. 1886. Each time after changing the respective settings, configuration files will be re-written.

If necessary, you can disable configuration files creation (see Section 9.1.4.5 on p. 1698).

When configuration files do not exist (for example, at the first launch of the KOMPAS-3D system), the internal default system settings are used.



If you need to start KOMPAS-3D with default settings, delete or rename configuration files.



The current configuration settings can be saved in the profile file (see Section 9.4.1 on p. 1871). Later on, using this profile will make it possible to quickly change the existing settings to the one saved in the profile.



Note that only those settings will be saved in *.cfg, *.prj and profile files that are different from the default ones. Information about unchanged parameters is not saved.

When reading *.cfg and *.prj files, or applying a profile to the parameters, about which there is no information, default values will be applied. The default path to SYS folder will be taken from KOMPAS.ini file (see Section 9.5.3 on p. 1883); if such file does not exist, the default path is determined using the value of the respective variable of KOMPAS environment (see Section 9.5.4 on p. 1886).

9.5.3. KOMPAS.ini file

KOMPAS.ini is a text file with data defining different system running operation parameters.



***KOMPAS.ini* is not included in the delivery suite.** You need to create it by yourself and place it to the same folder where *KOMPAS.exe* is located. *KOMPAS.ini* file is searched in the folder with the *KOMPAS.exe* file.

KOMPAS.ini can be created manually by clicking a special button in the system files location view dialog (see Section 9.1.4.1 on p. 1691).

Data in *KOMPAS.ini* is represented as values of the keys distributed over sections. Descriptions of sections and keys can be found in Table 9.5.2.

Tab. 9.5.2. Sections and keys of the KOMPAS.ini file

[Section]/key	Description
[Directories]	Contains paths to system files and folders.
Cfg	Specifies name of *.cfg configuration file and path to it *. Configuration files of libraries are also identified and located in the folder containing the *.cfg system file.

Tab. 9.5.2. Sections and keys of the KOMPAS.ini file

[Section]/key	Description
Prj	Specifies name of *.prj configuration file and path to it*.
Dsk	Specifies name of *.dsk configuration file and path to it*.
Temp	Indicates the folder to hold temporary files, as well as document** files which were automatically saved during a KOMPAS-3D session. During auto save parameters setup (see Section 9.1.4.4 on p. 1698), this folder will be offered as a system folder for temporary files.
Work	Specifies the folder to hold KOMPAS-documents**. This folder will be offered by default on the first time you open or save a document.
Sys	Specifies the folder where system files are located**: files of the user menus, user access, material density reference file, style libraries and so on. If the required system file is not found in this folder, the user will be able to specify the path to it. When setting up text templates, hole libraries and sketches, this is the default location folder of <i>Graphic.tdp</i> , <i>holelib.lfr</i> и <i>skchlib.lfr</i> files.
Libs	Specifies the folder where library files are located**. By default, this folder will be offered first when adding libraries to the Library Manager.
Templates	Specifies the folder where KOMPAS document template files** are located. When creating a document, the templates located in this folder shall be offered. The same folder will be offered by default to hold all newly created templates.
Profiles	Indicates the location folder of profile files**. When selecting profiles, those located in this folder will be offered. The same folder will be offered by default to hold all newly created profiles.
SheetMetal	Specifies the folder where bend table files are located**. When selecting bend tables, the tables located in this folder will be offered. The same folder will be offered by default when saving a bend table from a part into a file.
Thread	Specifies the folder where thread table files are located**. When selecting thread tables, the tables located in this folder will be offered.

Tab. 9.5.2. Sections and keys of the KOMPAS.ini file

[Section]/key	Description
MultilineTemplates	Specifies the folder where multiline template files are located ^{**} . When creating multilines, the templates located in this folder shall be offered. The same folder will be offered by default to hold all newly created templates.
PrintDevCfg	Indicates the location folder of printers' configuration files ^{**} . At saving of the configurations, this folder will be suggested as default. When loading configurations, those located in this folder will be offered.
Pictures	Indicates the folder containing the files with image examples that can be used for checking the part surface smoothness by the Environment reflection method ^{**} . This folder is offered by default for uploading images.
[Protection]	Contains information about specifics of data transfer to external device. This section only makes sense when the key is installed on the computer and an external device is connected through it.
PausePrinting	YES enables suspension of the output to the external device when KOMPAS-3D addresses the hardware protection key, NO — disables the suspension.
[Windows]	Contains information about the KOMPAS-3D logo during the system start-up.
ShowSplash	YES means that the logo will be displayed, NO — no logo is displayed.
[Bmp]	Contains information about the maximal block size for the output of bitmap and halftone images to printer.
MaxBlockSize	The maximal block size (in tens of kilobytes). For example, if the key is equal to 1000, bitmap and halftone images are output in blocks of 10 megabytes. If the key value is less or equals 0, all images are output in one pass.
[FontEscapement]	Contains information about the method of text output.
TextAsBitmap	YES enables text characters output in the form of a halftone, NO disables it. Enabling text output as a halftone makes it possible to solve the problem of printing system True Type fonts on some plotters (for example, Calcomp 720C).
[Files]	Contains paths to files with additional information.
ExprCollection	Indicates the name of the math expressions collection file <i>*.law</i> and path to it [*] .

Tab. 9.5.2. Sections and keys of the KOMPAS.ini file

[Section]/key	Description
GeneralTolerance	Specifies the name of the *.tfl file with unspecified maximum deviations of dimensions and a path to it. If the specified file does not exist, general tolerances for model dimensions are not assigned.

* If specified folder does not exist, it is created.

** If the specified folder does not exist, another folder is used. Path to the folder is defined as described in Section 9.5.4 on p. 1886.

To specify locations of folders and files in *KOMPAS.ini*, both absolute paths (for example, Sys=C:\Program Files\Ascon\KOMPAS-3D V...\Sys) and paths relative to the folder with the files *KOMPAS.ini* и *KOMPAS.exe* (for example, Sys=...\Sys) can be used.

Moreover, when setting paths, you can use Windows system variables (for example, Temp=%Temp%).

Lines starting with the semicolon (;) are considered commentaries and are not taken into account during *KOMPAS.ini* processing.

If the key's value contains a space, it should be enclosed in quotation marks (for example, Temp="D:\Working materials\Temporary files").

Sample *KOMPAS.ini* file:

```
[Directories]
Sys=..\Sys.
Libs=..\Libs
Templates=..\Templates
Cfg=%APPDATA%\Ascon\Kompas-3D\[version]tuning.cfg
Prj=%APPDATA%\Ascon\Kompas-3D\[version]new.prj
Dsk="%UserProfile%\Local Settings\Application Data\Ascon\Kompas-3D\[version]desktop.dsk"
Profiles="%AllUsersProfile%\Application Data\Ascon\Kompas-3D\[version]Profiles"
Temp=%Temp%
Work="%UserProfile%\My documents"
MultilineTemplates=..\Templates

[Windows]
ShowSplash=YES
```

9.5.4. Defining paths to system files and folders. KOMPAS-3D Environment Variables

Paths to system files and folders are defined during the KOMPAS-3D system startup and saved until the end of the session as values of the variable environment KOMPAS-3D. Names and val-

ues of the variables are the same as those of the keys of sections of the **[Directories]** and **[Files]** of *KOMPAS.ini* (see Table 9.5.2 on p. 1883).

In the paths to system files and folders below, the following parameters are used:

[APPDATA]—the folder containing information about applications for the current *User*;

[COMMON_APPDATA]—the folder containing information about applications and common for all PC users;

[version]—version number of the system in “X.X” format, for example: **10.0**;

[folder name]—the name of the system folder, the same as the variable name.

Depending of the version of the operating system, parameters **[COMMON_APPDATA]** and **[APPDATA]** accept values presented in Table 9.5.3).

Tab. 9.5.3. Values of the parameters **[COMMON_APPDATA]** and **[APPDATA]**

OS version	Parameter values
	[COMMON_APPDATA]
Windows XP	<i>C:\documents and settings\all users\application data</i>
Windows Vista, Windows 7, Windows 8	<i>C:\ProgramData</i>
	[APPDATA]
Windows XP	<i>C:\documents and settings\[user]\application data</i>
Windows Vista, Windows 7, Windows 8	<i>C:\Users\User\AppData\Roaming</i>

Defining paths to system files and folders:

1. Search of *KOMPAS.ini*, inside the file, paths to the system file (folder) are searched. If *KOMPAS.ini* is available and contains the required path, this path will be the value of the variable environment. *KOMPAS.ini* is described in detail in Section 9.5.3 on p. 1883.
2. If *KOMPAS.ini* does not exist or does not contain the required path, the path will be defined as follows:

- 2.1. For **Cfg**, **Prj**, **Dsk** variables:

Cfg=[APPDATA]\Ascon\Kompas-3D\[version]\Kompas.cfg

Prj=[APPDATA]\Ascon\Kompas-3D\[version]\Kompas.prj

Dsk=[APPDATA]\Ascon\Kompas-3D\[version]\Kompas.dsk

If the [APPDATA]\Ascon\Kompas-3D [version] does not exist, it will be created when the configuration file is saved.

- 2.2. For variables **SYS**, **LIBS**, **TEMPLATES**, **PROFILES**:

- ▼ Availability of the following folder is ensured:

[APPDATA]\Ascon\Kompas-3D [version]\[folder name].

If it is available,

[variable name]=[APPDATA]\Ascon\Kompas-3D\[version]\[folder name], for example:

- Sys=C:\Documents and Settings\sidorov\Application Data\ASCON\Kompas-3D\12.0\Sys.
- ▼ If this folder does not exist, availability of the following folder is ensured:
`[COMMON_APPDATA]\Ascon\Kompas-3D\[version]\[folder name]`.
 If it is available,
`[variable name]=[COMMON_APPDATA]\Ascon\Kompas-3D\[version]\[folder name]`,
 for example:
 Sys=C:\Documents and Settings\All Users\Application Data\ASCON\Kompas-3D\12.0\Sys.
 - ▼ If the specified folder does not exist, availability of the **[folder name]** is checked by the path relative to *KOMPAS.exe* (i.e. in the folder that contains the folder with *KOMPAS.exe*).
 If it is available,
[variable name]=[path to the folder containing the folder with KOMPAS.exe]\[folder name], for example:
 Sys=C:\Program Files\ASCON\KOMPAS-3D V13\Sys.
 - ▼ If the specified folder does not exist, the variable value is the path to the folder containing *KOMPAS.exe*, for example,
 Sys=C:\Program Files\ASCON\KOMPAS-3D V13\Bin.
- 2.3. For variable **MultilineTemplates**:
 The path is defined in the same way as for the **Templates** variable.
- 2.4. For variables **SheetMetal** and **Thread**:
- ▼ Availability of the following folder is ensured:
`[APPDATA]\Ascon\Kompas-3D\[version]\Sys\[folder name]`.
 If it is available,
[variable name]=[APPDATA]\Ascon\Kompas-3D\[version]\Sys\[folder name].
 - ▼ If this folder does not exist, availability of the following folder is ensured:
`[COMMON_APPDATA]\Ascon\Kompas-3D\[version]\Sys\[folder name]`.
 If it is available,
[variable name]=[COMMON_APPDATA]\Ascon\Kompas-3D\[version]\Sys\[folder name].
 - ▼ If the specified folder does not exist, availability of the `\Sys\[folder name]` is checked by the path relative to *KOMPAS.exe* (i.e. in the folder that contains the folder with *KOMPAS.exe*).
 If it is available,
[variable name]=.\Sys\[folder name].
 - ▼ If the specified folder does not exist, the variable value is the path to the folder containing the *Kompas.exe* file, for example,
- 2.5. For the **Temp** variable:
Temp=%Temp%
 If the variable of the **Temp (TMP)** environment is defined for the current user, this variable is used, for example,
 Temp="%UserProfile%\Local Settings\Temp".

Otherwise, the system variable **Temp (TMP)** is used, for example, `Temp=%System-Root%\Temp`.

2.6. For variables **Work** and **PrintDevCfg**:

Work="%UserProfile%\My documents",
PrintDevCfg="%UserProfile%\My documents".

2.7. For the **Pictures** variable:

▼ Availability of the `\Samples\Surfaces\Maps` folder by the path relative to *KOMPAS.exe* is checked (i.e. in the folder containing the folder with *KOMPAS.exe*).

If it is available,

Pictures=.\Samples\Surfaces\Maps.

▼ If this folder does not exist, then

Pictures="%UserProfile%\My documents".

2.8. For the **ExprCollection** variable:

ExprCollection=[APPDATA]\Ascon\Kompas-3D\[version]\collection.law.

2.9. For the **GeneralTolerance** variable:

▼ Availability of the following folder is ensured:

`[APPDATA]\Ascon\Kompas-3D\[version]\Sys`.

If it is available,

GeneralTolerance=[APPDATA]\Ascon\Kompas-3D\[version]\Sys\gen_tol.ttl.

▼ If this folder does not exist, availability of the following folder is ensured:

`[COMMON_APPDATA]\Ascon\Kompas-3D\[version]\Sys`.

If it is available,

GeneralTolerance=[COMMON_APPDATA]\Ascon\Kompas-3D\[version]\Sys\gen_tol.ttl.

▼ If the specified folder does not exist, availability of the `\Sys` folder is checked by the path relative to *KOMPAS.exe* (i.e. in the folder that contains the folder with *KOMPAS.exe*).

If it is available,

GeneralTolerance=.\Sys\gen_tol.ttl.

▼ If the specified folder does not exist, *gen_tol.ttl* is believed to be located in the same folder as *Kompas.exe*, for example:

`Sys=C:\Program Files\ASCON\KOMPAS-3D V15\Bin\gen_tol.ttl`.



Please, note that when *KOMPAS.ini* is missing, the file of unspecified maximum deviations can only have a default name — *gen_tol.ttl*. If the deviation file has to be named differently, use *KOMPAS.ini* to specify its name and location.

9.5.5. Service files

9.5.5.1. User Menu File

User menu file describes menus containing commands for insertion of text fragments to various texts. This file is named *Graphic.pmn* and located in subfolder \Sys of the KOMPAS-3D Main Folder.

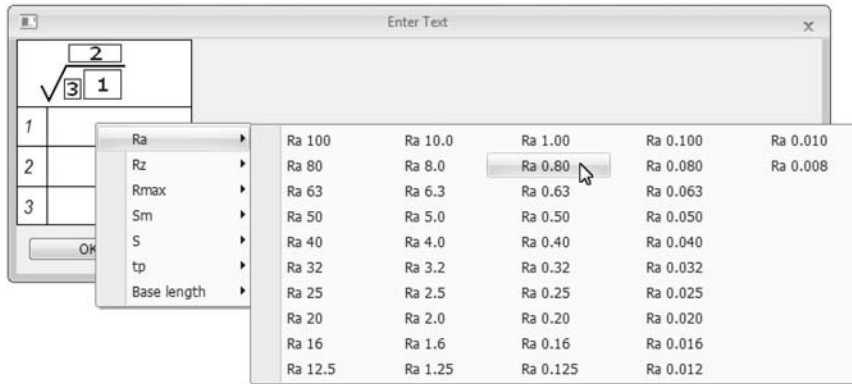


Fig. 9.5.1. User menu for text input in surface finish designation

Examples of menus described in the file *Graphic.pmn* are the menus called by double-clicking the left mouse button in the following cases:

- ▼ working with dialog for entering special character text (Fig.9.5.1), geometric tolerance, etc.
- ▼ filling the Document Title Block (Fig.9.5.2).

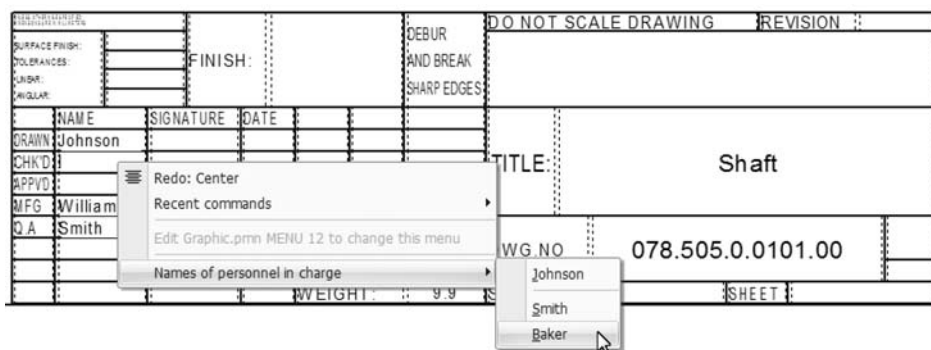


Fig. 9.5.2. User menu for filling the Drawing Title Block

Graphic.pmn is a text file that can be directly edited by the user.

By editing this file, the user can, for instance, customize the pop-up menu for filling the Title Block to include necessary names and initials of officials, complement the scale list, etc.

The user can also create new sections in *Graphic.pmn* file that describes menus available when entering a text into columns of original (user) Title Blocks (see Section 10.2.1.5.2 on p. 1968).



If the KOMPAS-3D System was been executing during *Graphics.pmn* file editing, then changes to the file will be available after KOMPAS-3D system is restarted.

9.5.5.1.1. File Structure

The file is comprised of sections. You can insert your own sections into the file. It is recommended to number them from 1001 to 4000.

Section numbers 1-1000 are reserved for ASCON. Please do not use them, otherwise your custom menus may conflict with new versions of *Graphic.pmn* file included to the system distribution package.

9.5.5.1.2. File Syntax

The User Menu File allows to build an hierarchic menu structure. It is comprised of individual lines. A line may describe a section or a menu command. Lines may be commented. Comments are not displayed on the screen or included in the document. Comment is a string preceded by ' (apostrophe) symbol. For example, in line **MENU 2 'Surface finish parameter** the comment is **Surface finish parameter**.

Each section of the file has a header **MENU num** where **num** is the section number.

A file section must begin with the keyword **BEGIN** and end with the keyword **END**. Lines located between those keywords (inside the section) will be displayed in the User Menu.

A section lists menu items (separate commands) and submenus (names of command groups) nested in this section. Submenus, in their turn, may also contain menu items and submenus.

A menu item has the following syntax:

MENUITEM "name" FLAGS, where

name is menu name,

FLAGS is flag combination (see Table). 9.5.7 on p. 1896).

A nested menu section, i.e., a submenu, has the following syntax:

POPUP "name", where

name is submenu name (which appears in the menu).

Similar to the parent menu, a submenu should begin with the keyword **BEGIN**, end with the keyword **END** and contain menu items (MENUITEM) and nested submenus (POPUP).

The syntax of **name** in a menu item or a submenu is as follows:

name = name1name2, where

name1 is a substring that will appear in the menu, i.e., command

name2 is a substring that will be inserted into the text when the command is selected.

If **name** consists of a single substring (of the form **name = name**), then this substring will appear in the menu and will be inserted into the text.

The substring **name1** may contain the control symbol &. The following symbol will be underlined and serve as shortcut key. A shortcut key allows to execute the command from the key-

board by pressing the appropriate key. The menu that contains this command should be displayed thereat on the screen. E.g., the file *Graphic.pmn* contains the string **MENUIITEM "&PolishPolish"**. In the User Menu, it will appear as **Polish**, and the P letter will serve as a shortcut key, i.e., after calling the menu containing the **Polish** command, for calling this command, it will be sufficient to press the *P* key on the keyboard.

The substring **name2** may contain various control symbols that define the configuration of a string inserted into the text. Various types of structures containing those control symbols and description of each structure are given below.

1. Insert Indices

The structure **\$XX;YY\$** allows to insert subscripts and superscripts in the document.

A substring entered after the first symbol **\$**, will be inserted in the document as a superscript. If the first symbol **\$** is followed by the symbol **;** then a substring after the symbol **;** will be inserted in the document as a subscript. The next symbol **\$** ends the sub/superscript. If the symbol **;** is missing or it is separated from the ending **\$** by an empty string, then only a superscript will be inserted. If the starting **\$** and **;** are separated by an empty string, then only a subscript will be inserted.

The menu item **AA\$XX;YY\$BB** will be inserted to the document as

AA^{XX}_{YY}BB

A menu item may contain several sub/superscripts. In this case the symbol **\$** that starts the sub/superscript must be separated from the symbol **\$** that ends the previous sub/superscript by at least one symbol. If a menu item contains the **\$\$** sequence, then a single symbol **\$** will be inserted in the document.

The symbols **s**, **m**, and **l** allow to change the size of sub/superscript symbols. They must follow the control symbol **\$**. Size-controlling symbols are active for the entire sub/superscript. Text will be small-sized after **s**, medium-sized after **m**, and full-sized after **l**. By default, a sub/superscript text is full-sized.

An example of entering a sub/superscript and changing its size is provided in Table. 9.5.4.

Tab. 9.5.4. Insertion of sub/superscripts into a document; changing size of a sub/superscript

Menu Item	Displayed Menu Command	String Inserted to Document	Index Height
MENUIITEM "L min cubed L\$3;min\$"			
MENUIITEM "L min cubed L\$l3;min\$"	L min cubed	L_{min}³	full
MENUIITEM "L min cubed L\$s3;min\$"	L min cubed	L_{min}³	small

2. Inserting Fractions

The structure **\$bXX;YY\$** or **\$dXX;YY\$** allows to insert a fraction in the document.

A substring entered after the symbol **\$b** or **\$d**, will be inserted to the document as a fraction numerator. If the symbol **\$b** (**\$d**) is followed by the symbol **;**, then a substring

after that symbol will be inserted to the document as a denominator of the fraction. The next symbol **\$** ends the fraction. If the symbol **;** is missing or it is separated from the ending **\$** by an empty string, then the fraction will have only a numerator. If the symbol **\$b (\$d)** is separated from the symbol **;** by an empty string, then the fraction will have only a denominator.

The menu item **AA\$dXX;YY\$BB** or **AA\$bXX;YY\$BB** will be inserted in the document as

AA $\frac{XX}{YY}$ BB

The symbols **s**, **m**, and **l** allow to change the size of fraction symbols. They must follow the control symbol **\$b (\$d)**. Size-controlling symbols are active for the entire fraction. Text will be small-sized after **s**, medium-sized after **m**, and full-sized after **l**. By default, fraction symbols are full-sized.

An example of entering fractions and changing their size is provided in Table. 9.5.5.

Tab. 9.5.5. Insertion of fractions into a document; changing size of a fraction.

Menu Item	Displayed Menu Command	String Inserted to Document	Fraction Height
MENUITEM "Clearance fit H7/h6 \$bH7;h6\$"	Clearance fit H7/h6	$\frac{H7}{h6}$	full
MENUITEM "Clearance fit H7/h6 \$bH7;h6\$"	Clearance fit H7/h6	$\frac{H7}{h6}$	small

A menu item may contain several fractions.

3. Insertion of Special Characters and Symbols

To insert special characters and symbols into structures, use the **~** (twiddle) symbol. It serves as a sign of the end of a special character number or a symbol code.

3.1. Insertion of KOMPAS System Special Characters

AA@+XXXXX~BB

A string of type

AA <special symbol N>XXXXX from Graphic.sss file> BB.

Numbers, images and names of KOMPAS-3D special characters are provided in the Appendix V..

E.g., the file *Graphic.pmn* contains the string

MENUITEM "2 holes dia.20|2 holes @+2~20".

In the User Menu, it appears as

2 holes dia. 20.

When it is selected, the following string will be inserted in the text:

2 holes ϕ 20

Thus, the structure **@+2** in the file *Graphic.pmn* substitutes the diameter symbol.

Special characters Nos. 78–80, 83, 9399–99, 171, 172 contain text (e.g., No.80 contains framed text). This text, in turn, may contain other special characters.

All symbols in the substring **name2** that follow the number of any special character that contains text are considered as composing the text of this special character (including control symbols).

E.g., the file *Graphic.pmn* contains the string

MENUITEM "Root of one third|@+98~\$b1;3\$".

In the User Menu, it appears as

Root of one third.

When it is selected, the following string will be inserted in the text:

$$\sqrt{\frac{1}{3}}$$


The largest permissible special character number is 65536.

If the number of a special character is greater than 65536, then the number shall be found as a remainder of the integer division of the specified number by 65536.

3.2. Insertion of Symbols from Fonts installed in the Operation System

AA^(FNAME)YXXXXX~BB

A string of type

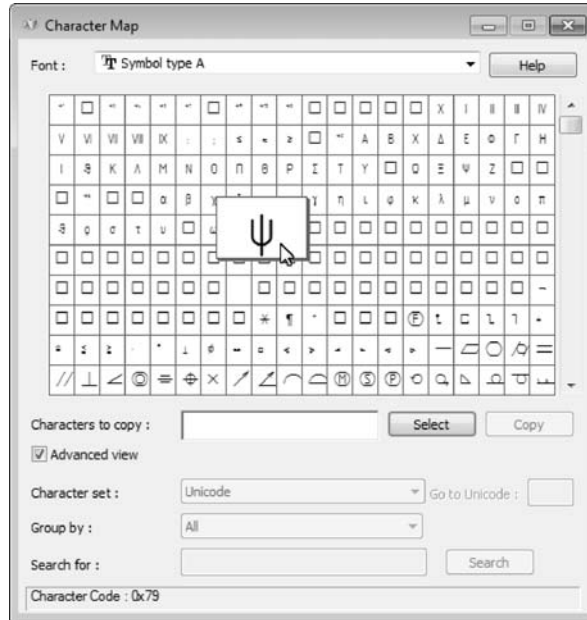
AA Symbol with code XXXXX from the font FNAME BB will be inserted to the text

The **Y** symbol in the **YXXXXX** structure designates a modifier of code representation. The modifier shows the numeral system, hexadecimal or decimal, in which the symbol code is represented (table. 9.5.6).

Tab. 9.5.6. Modifiers of symbol code representation

Modifier	Numeral system	Notes	Example
*	Hexadecimal system	Maximum number of symbols is 4. The case of A, B, C, D, E, F letters is not relevant.	*221E
+	Decimal system	Maximum number of symbols is 5. The largest code value is 65536; If the number of a special character is greater than 65536, then the number shall be found as a remainder of the integer division of the specified number by 65536.	+61448

Name of font style and codes of symbols may be determined, e.g., from standard Windows application **Character Map** (Fig.9.5.3).

Fig. 9.5.3. **Character Map** Application Window

E.g., the file *Graphic.pmn* contains the string

MENUITEM "Drum revolutions at max. deformation|^(Symbol Type A)+121~\$m*;3\$".

In the User Menu, it appears as

Drum revolutions at max. deformation.

When it is selected, the following string will be inserted in the text:

Ψ₃

3.3. Insertion of Symbols from Current Font

AA^YXXXX~BB

A string of type

AA Symbol with code XXXXX from the current text font BB will be inserted to the text

The Y symbol in the YXXXX structure designates a modifier of code representation. The modifier shows the numeral system, hexadecimal or decimal, in which the symbol code is represented (see table). 9.5.6).

To determine a style and a code of a symbol, you can, for example, use the standard Windows application **Character Map** (see Fig.9.5.3).



To insert the @, ^, ~, \$, & symbols from the current font in the text, you can use the @@, ^^, ~~, \$\$, && structures respectively, instead of the ^YXXXX~ structure.



Instead of basic structures for insertion of special characters and symbols i. e. **AA@YXXXX~BB**, **AA^(FNAME)YXXXX~BB** and **AA^YXXXX~BB**, you can use additional structures: **AA&XX~BB**, **AA#(FNAME)XX~BB** and **AA#XX~BB** respectively. Differences between basic and additional structures are the follows:

- ▼ you can use only two characters (not five) for designation of a special character number or a symbol code,
- ▼ modifier is not available in structures for insertion of symbols; a specified code is deemed to be decimal.

4. Insertion of a Line Break

AA@/BB

Two lines will be inserted to the text: **AA** and **BB**.

E.g., the file *Graphic.pmn* contains the string

MENUITEM "Block valve angular| Block valve@/angular".

In the User Menu, it appears as

Block valve angular.

When it is selected, two strings will be inserted in the document:

Block valve

Angular

Thus, the structure **@/** in the file *Graphic.pmn* works as a line break sign.



If a table cell of the Title Block is configured for single-line text, then any line breaks will be ignored.

Tab. 9.5.7. Flags for MENUITEM

Flag	Purpose
SEPARATOR	Draws a horizontal separator line. Any text or flags are ignored.
MENUBREAK	Places the menu command in a new column.
DISABLED or GRAYED	The menu command is unavailable for selection and displayed as grayed.

Flags may be combined, separated by spaces or commas.

Fragment of the file *Graphic.pmn* is provided below as an example of structure and syntax of this file.

MENU 12

BEGIN

MENUITEM "User-configured menu (Graphic.pmn MENU 12)" GRAYED

MENUITEM SEPARATOR

```

MENUITEM "I.V. &Nikolayeva   Vice President|Nikolayeva"
MENUITEM SEPARATOR
POPUP "Developers"
BEGIN
MENUITEM "P.P. &Polikarpov   Dept. Manager|Polikarpov"
MENUITEM SEPARATOR
MENUITEM "I.I. &Ivanov|Ivanov"
MENUITEM "P.P. P&etrov|Petrov"
MENUITEM "S.S. &Sidorov|Sidorov"
END
POPUP "Technologists"
BEGIN
MENUITEM "Yu.Yu. &Semenov   Dept. Manager|Semenov"
MENUITEM SEPARATOR
MENUITEM "A.B. &Dubinina|Dubinina"
MENUITEM "M.Ya. &Petrova|Petrova"
MENUITEM "I.I. P&olovin|Polovin"
END
POPUP "Quality Inspection"
BEGIN
MENUITEM "M.Yu. &Buyanov   Dept. Manager|Buyanov"
MENUITEM SEPARATOR
MENUITEM "P.P. &Polikarpov|Polikarpov"
MENUITEM "A.A. &Samokhvalov|Samokhvalov"
END
END

```

This section of the User Menu File describes the menu called when entering names of officials into fields of the Title Block (Fig.9.5.2 on p. 1890).

When editing the User Menu File, one should remember that some of its sections are used by the system for semi-automated text entry when setting dimensions and technologic symbols (for instance, a surface finish parameter value can be chosen from the User Menu). It is not recommended to edit those sections, as they contain standard texts and values.

9.5.5.2. Reference file for densities

In MP calculations of revolved and extruded solids, you can select a material density value from the special reference file instead of entering it manually.

The same file can be used for selection of material, replacement of material, and coverage at model properties setup. In this case, selection of the material makes it possible to define den-

sity and a preset hatch style (system or user-defined) to be displayed in cuts/sections of the drawings' associative views.

Names of materials and their respective density values, as well as links to hatch styles, are stored in the file *Graphic.dns* located in subfolder \Sys of KOMPAS-3D Main Folder.

Graphic.dns is a text file that can be directly edited by the user.

File sections begin and end with symbols «{» and «}» respectively. A section name must be on the same line as the opening brace. Sections can be nested. Spaces are not significant symbols and do not alter the appearance of strings in a reference dialog displayed on the screen.

Below is provided a fragment of the file *Graphic.dns* that describes the **Metals** section containing two subsections: **Aluminum Alloys** and **Bronzes**.

```
{Metals
{Aluminum Alloys
AD0 GOST 4784-97           = 2.71 | 0
AD1 GOST 4784-97           = 2.71 | 0
D1 GOST 4784-97           = 2.80 | 0
}
{Bronzes
BrAZh9-4 GOST 18175-78     = 7.50 | 0
BrAZhMts10-3-1.5 GOST 18175-78 = 7.50 | 0
BrKMts3-1 GOST 18175-78   = 8.40 | 0
}
```

The user can edit the file of densities, for example, by editing records in lines or by adding new ones.

A line of the density file is formed in accordance with the following rule:

<Material's designation> = *<density>* | *<file name>* | *<number>*

Material's designation is a record which is displayed in material's properties.

Density is a numeric density value in g/sm³.

File name is a name of a file from the library that contains hatch style. The following variants are possible:

- ▼ full name,
- ▼ short name, if the file is located in the \Sys subfolder of the KOMPAS-3D main folder,
- ▼ no file name, in case of the system style.

Number is a number of the style in the library of hatch styles or a system style number.

| — separator sign.

Below, you can find examples of lines with full file name, short file name, and without any name.

```
Polystyrene = 1.06 | d:\work\styles.lhs | 501
Polystyrene = 1.06 | graphic.lhs | 25
Polystyrene = 1.06 | 1
```

See Section 10.1.1.2.2 on p. 1912 for more information about creating custom hatch styles and saving them in the library.

List of hatch system styles and their numbers:

Metal	0
Non-metal	1
Natural stone	2
Ceramics	3
Concrete	4
Glass	5
Liquid	6
Natural soil	7
Made ground	8
Artificial stone	9
Armed concrete	10
Reinforced concrete	11
Wood in axial section	12
Wood	13
Sand	14

9.5.5.3. File of codes and names

The nomenclature of engineering documents is established by GOST 2.102–68. According to the standard, each document type has its certain code and naming. In KOMPAS-3D codes and names of non-substantive engineering, operational, and repair documents and diagrams are retrieved from the file *Graphic.kds*.

The contents of this file define the following:

- ▼ the content of **Codes and Names** dialog displayed after selecting the command for code and notation insertion,
- ▼ standard codes and names used
 - ▼ when auto-filling the BOM Title Block,
 - ▼ when disabling the display of product name in an item of BOM,
 - ▼ when synchronizing the Title Block of a drawing and a model.

The file *Graphic.kds* cannot be edited in KOMPAS-3D. However, this is a text file and thus it can be opened and edited in any text editor, e.g., in Windows Notepad.

File sections begin and end with symbols «{» and «}» respectively. A section name must be on the same line as the opening brace. Sections can be nested. This allows to construct a tree-like structure of document types. Spaces are not significant symbols and do not alter the appearance of strings in a dialog.

Below is provided a fragment of the file *Graphic.kds* that describes the *Repair Documents* section of **Codes and Names** dialog (Fig.9.5.4). This section contains the following subsections:

- ▼ **Major Repair;**
- ▼ **Medium repair.**

The example shows the correspondence between structure and contents of *Graphic.kds* file and those of the dialog.

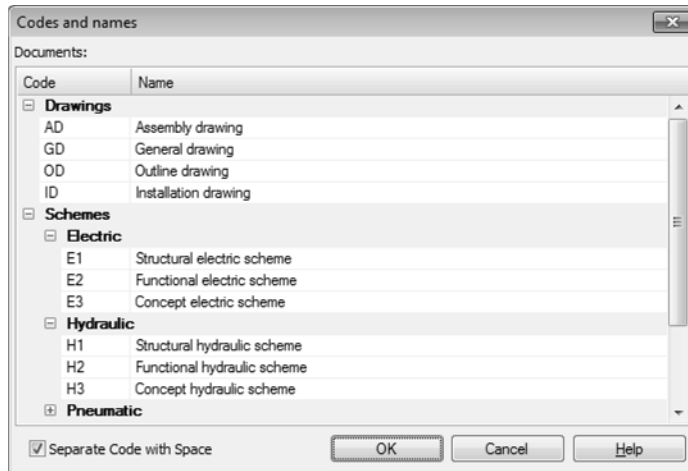


Fig. 9.5.4. **Codes and Names** dialog; Repair Documents section

{Repair documents

{Major Repair

RK = Repair Manual

UK = Repair Specifications

ZK = Norms of spare parts usage for repair

MK = Norms of material usage for repair

ZIK = Schedule of spare parts for repair

VRK = Schedule of repair documents

}

{Medium repair

RS = Repair Manual

US = Repair Specifications

ZS = Norms of spare parts usage for repair

MS = Norms of material usage for repair

ZIS = Schedule of spare parts for repair

VRS = Schedule of repair documents

}

The file *Graphic.kds* should be located in a folder with the path defined by the system variable *SYS*. By default, this is \Sys folder of KOMPAS-3D System.

The system checks the presence of the codes and names file in the following cases:

- ▼ when calling the command for codes and names insertion,
- ▼ when creating a BOM item in a drawing with completed Title Block,
- ▼ when switching to the slave mode of work with BOM items in a drawing with completed Title Block,
- ▼ when opening a drawing with completed Title Block and with BOM Table in drawing,
- ▼ when opening a BOM that contains at least one object and the completed Title Block,
- ▼ when opening a BOM with the Title Block that is associated with a document attached to it.



The availability of the codes and names file is checked once per working session, i.e., in one of the above list cases that occurs first.

If the file *Graphic.kds* is missing in the folder that was set as default, then the system displays the dialog shown in Figure 9.5.5.

Buttons of this dialog are described in the Table 9.5.8.

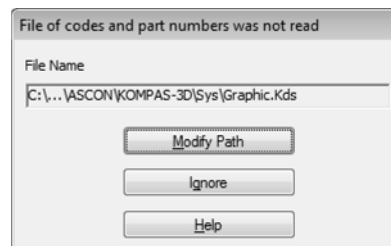


Fig. 9.5.5. Code and Name File Replacement Dialog

Tab. 9.5.8. Buttons of Codes and Names file replacement dialog

The	Function
Change Path	Searching for the codes and names file on the disk. Upon pressing this button, the standard Windows Open File dialog will be displayed. You can select any file (also, with an extension other than *.kds), whose structure matches that of a codes and names' file. The selected file is used only for the current working session. After the next start, the system will again search for the file <i>Graphic.kds</i> in the system folder defined by the variable <i>SYS</i> .
Ignore	Abandon the use of the file <i>Graphic.kds</i> in the current working session. After pressing this button, automated entry and processing of codes and names when working with a BOM will become unavailable until the end of the session.

9.5.5.4. Library Manager Configuration File

Library Manager configuration files contain the description of the Library Manager structure. Their file extension is *lms*. These files are used in the following cases:

- ▼ initial populating of the Library Manager,
 - ▼ execution of **Update Library Manager** command.
- Upon launching the Library manager or executing **Update Library Manager** command, the following actions will be performed.
- ▼ Analysis of the configuration file.
 - ▼ Checking that library files listed in the configuration file are present in the folder *\Libs* of the KOMPAS System.
 - ▼ Adding the files found to the Library Manager according to the specified structure.

If before the **Update Library Manager** command execution the Manager contains libraries not listed in the configuration file, they remain unchanged. If the Manager contains libraries listed in the configuration file, but with paths inconsistent with this file, then their location in the Manager will not change. Libraries are not added repeatedly.

The configuration file cannot be edited in KOMPAS-3D. However, this is a text file and thus it can be opened and edited in any text editor, e.g., in Windows Notepad.

There may be several configuration files. The Library Manager is populated using contents of all configuration files. User configuration files for the Library Manager should be assigned the extension *lms* and stored in the folder *\Sys* of the KOMPAS System.

Every line of the file contains a description of an attached library and includes the following parts:

- ▼ Path to the library in the Manager,
- ▼ Name of the Library File,
- ▼ Comment.

Parts of the file line are preceded by the following notations:

- ▼ */d-* path to the library in the Manager, path elements are separated by vertical bar |,
- ▼ */f-* library file name,
- ▼ */c-* comment.

Below is provided a fragment of the file *Graphic.lms* included to the KOMPAS-3D distribution kit.

```
/d-Sample Libraries\Symbols/f-Graphic.lfr/c-Design Element Library
```

```
/d-Sample Libraries\Symbols/f-Tech.lfr/c-Technologic Symbol Library
```

An example of Library Manager structure built in accordance with this fragment is depicted in Fig.9.5.6.



Fig. 9.5.6. Example of Library Manager structure

9.5.6. Recommended settings of the KOMPAS-3D system during group work

This section contains some recommendations on organizing a group work of users of the KOMPAS-3D system. Before using the recommendations, you should read Sections 9.5.1 – 9.5.4.

KOMPAS-3D Using these recommendations makes it possible to organize working with the system so that the following requirements are met.

- ▼ Common design of all issued documents (paper and electronic): some enterprises have their own standards with respect to documentation, corporate templates, forms and so on.
- ▼ Centralized documentation storage.

To set up KOMPAS-3D system for group work, do the following:

1. On a network computer (on the server, for example), organize a shared resource. It should be divided into four parts:
 - ▼ folder for original documents,
 - ▼ folder for backup document copies,
 - ▼ folder for configuration files,
 - ▼ folder for system files.
2. Connect the created shared resource to KOMPAS-3D workplaces as a **network disk**— this will make it simpler to change the resource location, if necessary.
3. From a workplace, set up the KOMPAS-3D system as required. Pay special attention to the following objects of the setup.
 - ▼ Setting access level (see Section 9.1.4.2 on p. 1694).
 - ▼ Backup copying (see Section 9.1.4.3 on p. 1695). To save the copies, specify the subfolders of the folder for backup copies on the shared resource.

The system settings will be saved in the **.cfg* file. Its default name is *KOMPAS.cfg*.

4. Move the **.cfg* file with the settings made to the folder with the configuration files located on the shared resource.
5. On a KOMPAS-3D workplace, setup document design, styles, text templates, user menus and other system files that are included in the supply kit (by default, all these files are located in the \SYS subfolder of the main system folder). If necessary, you can create your own libraries of templates and styles.

6. Drag the contents of the \SYS folder (including files edited or created by you) to the system files folder on the shared resource.
7. Set up new documents, subject to document requirements in your company.
New document settings will be saved in the *.prj file. By default, its name is *KOMPAS.proj*.



Settings listed in paragraphs 5 and 7 require certain experience in working with the KOMPAS system.

8. Move the *.prj file with the settings made to the folder with the configuration files located on the shared resource.
9. Create the following file: *KOMPAS.ini* with the **[Directories]** section.
 - 9.1. In the **CFG** key, specify the path to the *.cfg file.
 - 9.1. In the **PRJ** key, specify the path to the *.prj file.
 - 9.2. In the **SYS** key, specify the path for the system files.
 - 9.3. In the **WORK** key, specify the path for the original documents.
10. Copy the resulting *KOMPAS.ini* file to each KOMPAS workplace and place it in the same folder with the *KOMPAS.exe* file.

Similar document design when using the described settings is achieved by means of the following:

- ▼ settings for new documents will be the same at all workplaces, since these parameters are taken from one *.prj file,
- ▼ document design, line drawing, text, hatches, view and contents of template titles and other standard objects will be identical, since the same system files are used,
- ▼ thickness of lines in paper documents, as well as the objects printed, are the same, since the respective information is taken from one *.cfg. file



For uniform document design, you can use **templates of KOMPAS documents**.

When the required templates are created (or those included in the supply kit are edited), put them into a separate file on the shared resource. To sort multiple templates and search through them conveniently, subfolders can be created in the folder.

In the **TEMPLATES** key of section **[Directories]** of the *KOMPAS.ini* file, specify the path to the templates folder.

Centralized document storage is archived, if the described setting is used, thanks to the fact that when documents are opened and saved at any workplace, the same folder is automatically offered for the saving— the original documents folder.



To facilitate managing documents (search, archiving and so on), we recommend storing them in a sorted manner. You can arrange the documents using different files for different projects and so on.

10.Styles of Objects. Document Layouts

10.1. Object styles

10.1.1. General Information

External view of an item (a line, a point, a hatch or a text fragment) is defined by its **style**.

A KOMPAS-3D system is supplied with ready-made styles which may be used when creating and editing objects.

Custom styles can be created.

Custom styles can be stored (see also Table 10.1.1):

- ▼ Directly inside a **document**
- ▼ In external files — **style** libraries,
- ▼ In the named groups — **sets**.

10.1.1.1. Types of Styles and their Storage

Styles of points, lines, hatches and texts saved directly in a program code are called **system** styles. Additionally, the scope of delivery includes a line style library (*Graphic.lcs*) and a hatch style library (*Graphic.lhs*).

A user can create his own line, hatch and text styles. Custom point styles cannot be created.

When creating new styles, a user can save them in different locations. Possible locations for style storage and special features of their operation are represented in Table 10.1.1.

Tab. 10.1.1. Style storage locations

Storage location	Operational features
Document	Styles saved directly in a document file (embedded in the document styles) are transferred with it to other workstations. These styles belong to this document and are available only in it. To use styles in other documents, you must, at first, copy them from a document into a set or a library.
Library	Style libraries — the files external to documents [*] . If you transfer the documents, which use library styles, to other workstations, you must also transfer style libraries. Styles saved in libraries are available when different documents are created. Edition of a library style results in change of the view of all objects which use this style.

Tab. 10.1.1. Style storage locations

Storage location	Operational features
Set	<p>A set is a named group of styles. The information on user-created sets is stored in the file with the extension <i>*.cfg</i> (see Section 9.5.2 on p. 1881).</p> <p>By default, the styles contained in sets are available only at the workstation where they were created**.</p> <p>The advantage of style sets over style libraries consists in the following. The system remembers a once-used set until the end of a working session, while you have to refer to a library every time you need it. Thus, it is easier to access set styles again than to access library styles.</p>

* The files of libraries containing line styles use the extension *lcs*, hatch styles, *lhs*, and text styles, *lts*.

** Style sets can be transferred from one workstation to another by moving a **.cfg* file. Please note that, in addition to the data on sets, this file contains the settings of the KOMPAS-3D system.

10.1.1.2. Operation of Styles

Line, hatch and text styles are operated in dialogs activated by the commands **Service — Li-**

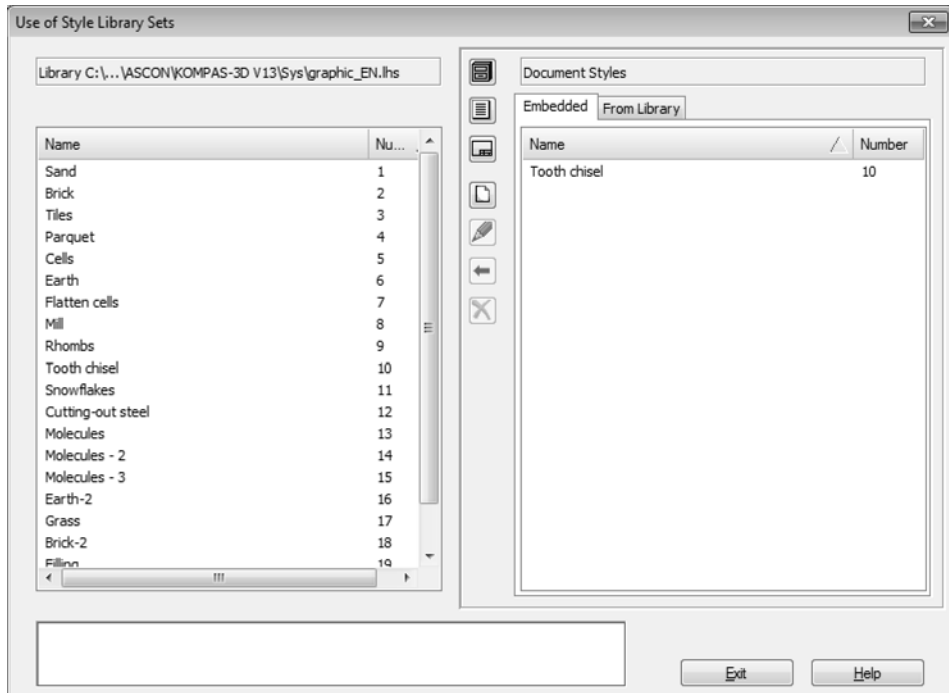







Fig. 10.1.1. Use of Hatch Style Libraries and Sets dialog



braries of Styles, Layout Types and Title Blocks — Line Styles... /.Hatch Styles... / Text Styles.... For example, on the Fig. 10.1.1 is shown use of Hatch Style Library Sets dialog.

The dialogs have the same control elements. They are represented in Table 10.1.2.

Tab. 10.1.2. Use of Style Libraries and Sets dialog

Element	Description	
Preview Window	<p>The window in which a selected style set is displayed. Since two preview windows are available, two different lists of styles can be displayed in a dialog at the same time. Only one window can be active it is selected by a marquee. For example, in Fig. 10.1.1, the right window is active. To activate, click the mouse in the window. Styles can be sorted in the lists by names or by IDs. To alter list sorting sequence click the desired column header with the mouse. A triangle mark will be displayed in the header of the column used for sorting styles. Its position shows the sorting direction: ascending or descending.</p> <p>If a list of document styles or a list of set styles is selected for display, the preview window contains two pages.</p> <p>On the Embedded page a list of styles saved directly in a document or in a set is displayed (see Fig. 10.1.1). On the Library page, a list of styles saved as links to respective library styles is displayed.</p>	
	Show Set	<p>Makes it possible to display a list of styles, which are saved in a set, in an active window. After clicking this button, a dialog with a list of earlier created style sets appears on the screen. You must select a name of a required set in this dialog.</p>
	Show Library	<p>Makes it possible to display a list of styles, which are saved in a library, in an active window. After clicking this button, a screen displays the dialog, in which you must specify the name for a style library file to be opened.</p>
	Show document	<p>Makes it possible to display a list of document styles in an active window. After clicking it, a list of styles, saved or used in a current document, appears in an active window. If there are no opened documents and if the current document cannot contain the objects, the styles of which are operated, this button is not available. For example, if a text document is active, the Show Document button will be not available in Line Style and Hatch Style Operation dialogs.</p>
	New	<p>The button allowing creation of a new style. After clicking it, a Style Creation and Edition dialog appears on the screen. A newly created style will be saved in a document (set, library), a list of styles of which is displayed in an active preview window.</p>
	Edit	<p>The button making it possible to edit a selected style. After clicking it, a Style Creation and Edition dialog appears on the screen.</p>

Tab. 10.1.2. Use of Style Libraries and Sets dialog

	Element	Description
	Copy	<p>The button making it possible to exchange styles between libraries, sets and documents. After clicking it, the selected style (styles) is (are) copied to the neighboring preview window.</p> <p>If, in the process of copying to a document or a set, the Embedded page is active, the style is copied immediately to a document or a set.</p> <p>If, in the process of copying to a document or a set, the Library page is active, a link to a style to be copied is created in a document or a set.</p>
	Delete	<p>A button making it possible to delete a selected style (selected styles). The detailed information on deleting styles is given in Section 10.1.1.2.3 on p. 1914.</p>

To finish the operation with styles, use the **Exit** button. After clicking it, all additions, deletions or changes will be automatically saved.

10.1.1.2.1. Creating Libraries and Style Sets

To create a new style library, complete the following steps.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Line Styles... / Hatch Styles... / Text Styles...** depending on the type of objects whose styles you want to work with.

A Style Set and Style Library Operation dialog will appear on the screen.



2. Click button **Show Library**.
3. In the dialog that appears on the screen, select or create a folder in which a new library will be located.
4. Enter a name of a new (non-existing) library and click the **Open** button.
5. Respond **Yes** to a system prompt for creation of a new file (Fig.10.1.2).

The active preview window of a Style Operation dialog will be cleared since the newly created library is empty.

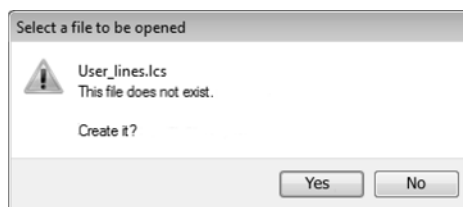


Fig. 10.1.2. Prompt for creation of a new library file



You can practice the technique of creating a style library by doing Exercise 1 on p. 1924.

To create a new set of styles, complete the following steps.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Line Styles... / Hatch Styles... / Text Styles...** depending on the type of objects whose styles you want to work with.



2. Click the **Show Set** button.



3. In the appeared Style Set Operation dialog click the **New...** button.

4. In the Name Set dialog, which appears on the screen, enter a set name and click the **OK** button.

Both dialogs will be closed, while the Style Operation dialog will remain on screen. The active preview window for the Style Operation dialog will be cleared since the newly created set is empty.

Using the **Create** button, you can create new styles in a library or in a set.



Using the **Copy** button, you can copy into a library or a set already existing styles. To do this, in a neighboring window you must open a required source of styles— a library, a set or a document — for copying.



10.1.1.2.2. General Technique of Creating Styles

To create a new style, complete the following steps.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Line Styles... / Hatch Styles... / Text Styles...** depending on the type of objects whose styles you want to work with.



A Style Set and Style Library Operation dialog will appear on the screen.

2. Select a new style storage location.



To achieve this, use the **Show Library...**, **Show Set...**, or **Show Document** buttons. You can also create a library or a set (see Section 10.1.1.2.1) for location of a new style.



3. Click the **Create** button.



Note that a new style can be created in the current document or set if the **Embedded** page is active. After clicking the **Create** button, an Object Style Creation and Editing dialog will appear on the screen. Operation with a Line Style Creation dialog is described in Section 10.1.2.1 on p. 1920, with a Hatch Style Creation dialog — in Section 10.1.3.1 on p. 1931, and with a Text Style Creation dialog — in Section 10.1.4.1 on p. 1947.

4. Customize properties of a new style and close the Style Creation dialog using the **OK** button.

The name of the style created will appear in an active preview window of the Style Set and Style Library Operation dialog. You can edit a new style or copy it to another location — to a library, a set or the current document.

A new style of the line, hatches or text in the **current document** can be created during creation or editing of an object — line, hatches or text.

When creating or editing a line, the following actions shall be done to create a new line style:

1. Unfold the **Style** list on the Property Bar and select the **Another Style** string.

The dialog to select the line style will be displayed on the screen (Fig.10.1.3).

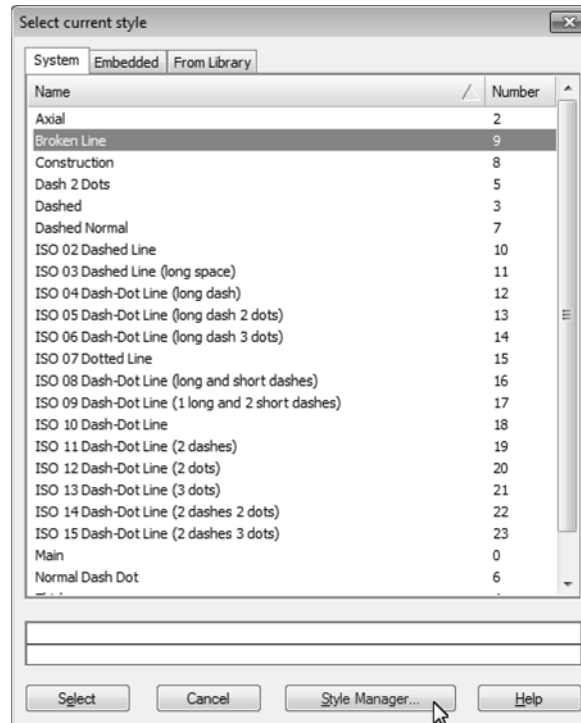


Fig. 10.1.3. Select Line Style dialog

2. Click the **Style Manager...** button in the dialog.

The Line Style Manager dialog will appear on the screen (Fig.10.1.9).



3. Click the **New Style** button in the Line Style Manager.

The dialog for creating and editing named groups will appear. Work with these options is described in Section 10.1.2.1 on p. 1920.

4. Customize properties of a new style and close the Style Creation dialog using the **OK** button.
5. The name of the created style will appear in the Line Style Manager list. Close the Line Style Manager by clicking **OK**.

The name of the created style will appear in the style selection dialog on the **Embedded** page. You can set this style for the current style by clicking the **Select** button.

When creating or editing the hatches (text), the following actions shall be done to create a new hatches (text) style:

1. Unfold the **Style** list on the Property Bar and select the **Another Style** string.

The style selection dialog will appear. An example of the hatches style selection dialog is shown in Fig.10.1.4; the text style selection dialog is analogous to the shown one. By default, a list of document styles is displayed in a Style Selection dialog (which is indicated by a line in the upper part of the dialog — see Fig.10.1.4).

2. If this is not the case, click the **Document** button in the **Show** group.
3. Activate the **Embedded** page and click the **New...** button.

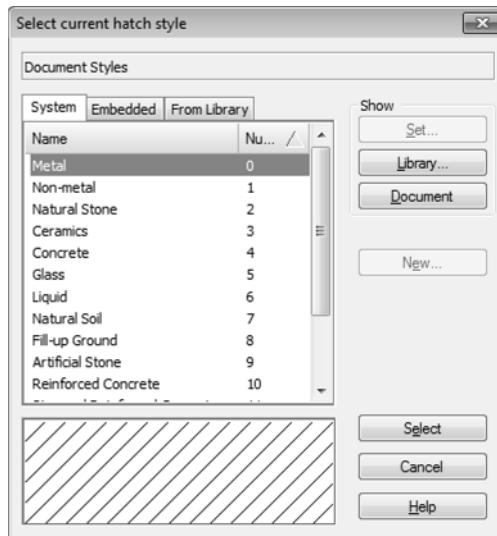


Fig. 10.1.4. Select Hatch Style dialog

The dialog for creating and editing a style object will appear. Work with the hatches style creation dialog is described in Section 10.1.3.1 on p. 1931, and work with the text style creation dialog — in Section 10.1.4.1 on p. 1947.

4. Customize properties of a new style and close the Style Creation dialog using the **OK** button.

The name of the created style will appear in a list on the **Embedded** page. You can set this style for a current object by clicking the **Select** button.

A style created when creating or editing an object is no different from a style created in the dialog of working with style sets and libraries.

10.1.1.2.3. Deleting Separate Styles, Style Sets and Style Libraries

To delete the styles which have become unnecessary, perform the following operations.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Line Styles... / Hatch Styles... / Text Styles...** depending on the type of objects whose styles you want to work with.

A Style Set and Style Library Operation dialog will appear on the screen.

2. In the dialog, specify the style library that stores the style to be deleted.

For this purpose, use **Show library...**, **Show set...**, or **Show document** buttons.

3. Select the styles to be deleted and click the **Delete** button.

A Style Delete dialog will appear on the screen (Fig.10.1.5). This dialog contains the name of the style to be deleted and control buttons for deleting. The functions of the buttons are represented in Table 10.1.3.



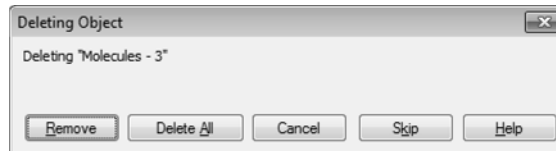


Fig. 10.1.5. Style Deletion dialog

Tab. 10.1.3. Control buttons for deleting object styles

The	Purpose
Delete	Deleting the style whose name is displayed in the dialog. After clicking this button, you proceed to the next style (if multiple styles were selected for the deletion).
Delete all*	Deleting all selected styles.
Skip *	Undoing the style deletion, whose name is displayed in the dialog. After clicking this button, you proceed to the next style.
Cancel	Interrupting the style deletion operation.

* The button is available if several styles are selected for deletion.



Line styles embedded in the document can be deleted in the Line Style Manager (see Fig.10.1.9) provided they are not used.

Be careful when deleting styles, as their recovery is impossible.

To delete a style set which has become unnecessary, perform the following operations.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Line Styles... / Hatch Styles... / Text Styles...** depending on the type of objects whose styles you want to work with.
A Style Set and Style Library Operation dialog will appear on the screen.
2. Click the **Show Set...** button.
3. In the Style Set Operation dialog, which appears on the screen, select a set to be deleted and click the **Delete** button.
4. Respond **Yes** to the system prompt for deletion of a set.
5. Click the **Exit** button of a Style Set Operation dialog.



Deletion of the current set (a list of styles of which is at the moment displayed in the Style Set and Style Library Operation dialog) is impossible.

Be careful when deleting sets: after logging off the Set Operation dialog, cancellation of the operation is impossible. You can recover a set only immediately after a positive response to the prompt for deletion. To do it, click the **Undo** button in the Style Set Operation dialog.

To delete a style library, you must delete its file from the disk.

10.1.1.2.4. Number of a Style

Each style has an index number which is assigned to it when this style is created.

The index numbers of styles saved in documents and sets can coincide, as these styles are used directly (without making any links).

The index numbers of styles saved in a library must differ. This is connected with the fact that when a library style is assigned to an object in a document that contains this object, a link to this style appears. The link includes the following data: a full library name and an index number of a style from it.

If you try to insert a style into a library (by creating or copying) and the index number of this style is the same as that of one of the styles already existing in this library, a Warning dialog appears on the screen (Fig. 10.1.6).



Fig. 10.1.6. Warning dialog about the presence of a style in a library

It contains the names of styles having the same index numbers, and control buttons. The functions of the buttons are represented in Table 10.1.4.

Tab. 10.1.4. Buttons controlling the insertion of an object into a library

The	Purpose
Add while saving the old one	Inserting an object into a library without deleting the existing object from it. The first of the available (“vacant”) numbers will be assigned to a new object.
Add as replacement of an old one	Inserting an object into a library with replacement of the existing object. The existing object will be deleted— thus, its index number will become available for use. A new object will be inserted into a library without any change of its index number.

Tab. 10.1.4. Buttons controlling the insertion of an object into a library

The	Purpose
Do not Add	Cancel insertion of an object into a library. To cancel insertion of a newly created object into a library means the loss of all settings made. Since in this case an object to be inserted is not used in any document yet, it should be suitably added (to the library), saving the existing object.



Should a library have a style, the index number and the name of which is the same as those of the style to be added, a prompt to confirm rewriting the style— replacement of the old style with a new one — will appear on the screen. You can confirm or reject the overwrite.

If there is a rather high probability of style interchange between several libraries, you must provide different index numbers to all styles in these libraries. It will help to prevent the conflict situations described above.

10.1.1.2.5. Storing the Data on Object Styles in a Document

Thus, if an item (a line, a hatch or a text) uses a style different from the system's, two options for storing data on this style in a document are available.

The first option— storing the style itself inside a document (embedded style).

The second option— storing a link to a library that contains the style used (library style).



All links available in a current document, including the links to style libraries, can be viewed on the **External Links** page of the Document Data dialog. Use the **File— Document summary...** command to activate this dialog.

Both sets and documents can contain either styles (their list is displayed on the **Embedded** page— see Fig.10.1.1 on p. 1909), or links to library styles (their list is displayed on the **Library** page).

In case a style that is directly contained in a set is assigned for an object, the used style is copied into a document, i.e., the first option of storing data on a style is employed.

In case a style that is contained in a set as a link, is assigned for an object, a link to a selected style is also made in a document, i.e., the second option of saving data on a style is employed.

Editing the style saved in a document results in changes in this document only: the appearance of all objects that use the edited style changes.

Editing the style saved in a library results in changes in all documents containing the objects that use the edited style.

If the document is opened and the library, the hatches or text style of which is used in this document, is not found (is deleted, moved or renamed), then a Warning dialog appears on the screen (Fig.10.1.7).

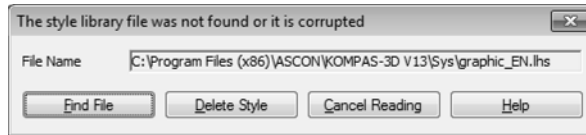


Fig. 10.1.7. Warning dialog notifying of the absence of a hatches style library

The dialog contains the full name of the library not found and control buttons. The functions of the buttons are represented in Table 10.1.5.

Tab. 10.1.5. The buttons of a Warning dialog notifying of the absence of a style library

The	Purpose
Find File	Selecting the library file from which a style must be taken.
Delete style	Deleting a link to a library and assigning a system style to objects.
Cancel Reading	Cancellation of the command to open a document.

If the library to which a document refers is found but it lacks the hatches or text style with a required index number, then a style is replaced according to the following rule:

- ▼ System style *Metal* is used in stead of the hatch style not found.
- ▼ Instead of the text style not found — a corresponding system style (*Text on Drawing*, *Dimension Texts*, etc.).

If at the moment of file opening a library, from which the line style is used for this document, is not found, or the style is not found in the library, then a message shown on Fig.10.1.8 will appear on the screen.

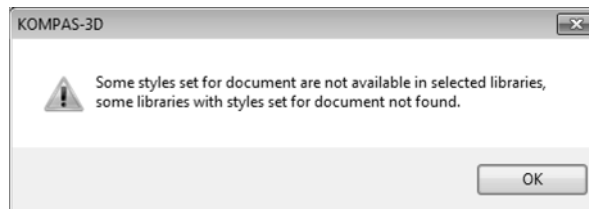


Fig. 10.1.8. Warning dialog notifying of the absence of libraries or line styles.

Drawing of a line using a non found library style does not change.

10.1.1.3. Peculiarity of work with line styles

Work with line styles (in contrast to the work with hatches and text styles) is characterized with a possibility of generation of a list of styles of the current and new documents.

The list setting up is carried out in the Line Style Manager (Fig.10.1.9). When working with the current document, the Manager also allows to carry out main actions with the line styles — creation, editing and deletion.

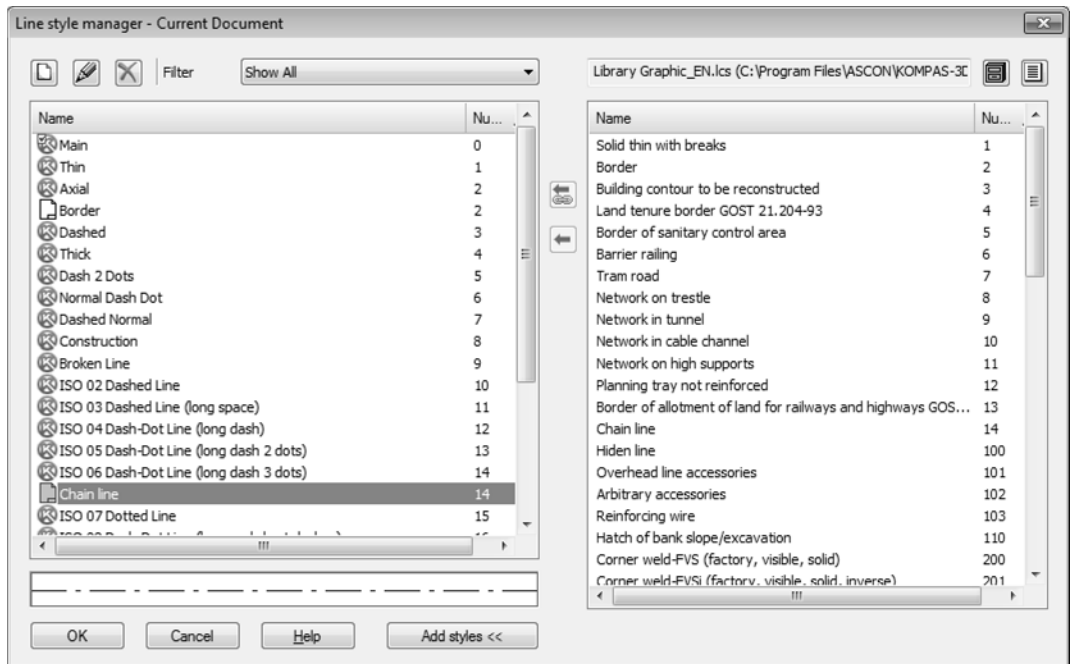


Fig. 10.1.9. Line Style Manager during the line style list setting up for the current document

The document style list is displayed in the left part of the Line Style Manager. The buttons located above the list allow to do the following:



- ▼ Create new line styles in a document



- ▼ Edit the embedded styles



- ▼ Delete the embedded styles (provided that they are not used).

A library (or a set) can be opened in the right part of the Line Style Manager (after clicking the **Add Styles** button) for inclusion of styles to the list of document styles. Two methods of style addition are available:



- ▼ Creation of a link to a library style



- ▼ embedding a style from a library or a set into a document.

The Line Style Manager can be called by two ways:

- ▼ By clicking the **Modify List...** button in the line style setting up dialog for the current or new graphics documents
- ▼ By clicking of the **Style Manager...** button in the line style selection dialog (see Fig.10.1.3).

10.1.2. Custom Line Style

To create a line style, perform the following operations.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Line Styles...**



A Style Set and Style Library Operation dialog will appear on the screen.



2. Select a new style storage location.

To achieve this, use the **Show Library...**, **Show Set...**, or **Show Document** buttons. Style storage locations are described in Section 10.1.1.1 on p. 1907.



3. Click the **New** button.

The dialog for creating and editing named groups will appear.



4. Customize the parameters of a new style (see Section 10.1.2.1) and close the dialog.

The name of the style created will appear in an active preview window of the Style Set and Style Library Operation dialog.



After reviewing the material set forth in Section 10.1.2.1, it is recommended to do the exercises of Section 10.1.2.2. It will allow you to practice the described technique of creating line styles.

10.1.2.1. Customizing a Line Style

Customizing the parameters of a line style is performed in a Style Creation and Edition dialog (Fig. 10.1.10). The control elements of this dialog are represented in Table 10.1.6.

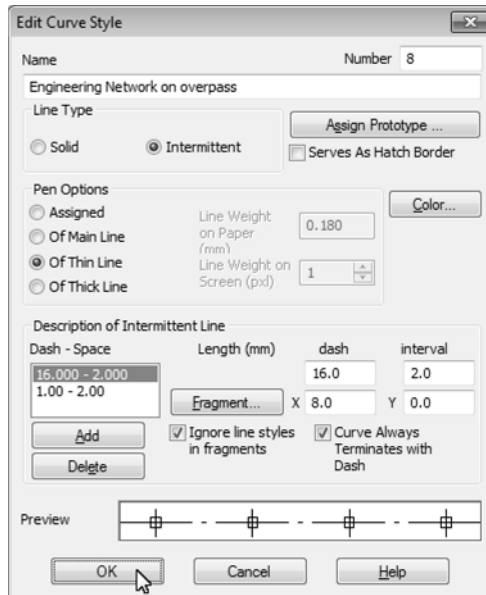


Fig. 10.1.10. Dialog to Create or Edit Line Style

Tab. 10.1.6. Dialog to Create or Edit Line Style

Element	Description
Name	A field for entering (editing) a style name. It is recommended that informative style names be given which fairly represent the function of such styles. In the future this will simplify the search for a style in a list.
Number	A field for entering (editing) a style number. The rules of style numbering are set forth in Section 10.1.1.2.4 on p. 1916.
Curve Type	A group of options making it possible to select a line type: Solid or Intermittent .
Assign Prototype	A button making it possible to select a current style as a template (a prototype) of a newly created style. Assignment of a prototype is useful if a new style is a modification of that already existing style. After clicking this button, a dialog, in which you must select a prototype style, will appear on the screen. Thereafter, options are activated and values corresponding to the properties of a prototype style appear in the Line Style Creation and Edition dialog.
There appears a border for hatch	Activation of this option means that the lines of this style will be accounted for by the system in the process of auto determination of hatch borders.
Pen Settings	Group of elements (see Table 10.1.7), where parameters of lines appearance on the screen and in print can be set up.
Color	A button allowing selection of a line color.
Description of Intermittent Curve	Group of elements (see Table 10.1.8) for setting the intermittent curve parameters. This group is available if in the Line Type group, the Dashed option is activated.
Preview	This field shows the appearance of the line style to let you immediately evaluate the changes you made.

Tab. 10.1.7. Elements of the group **Pen Options**

Element	Description
Assigned	An option making it possible to set arbitrary pen options. After its activation, the fields for entering width become accessible.

Tab. 10.1.7. Elements of the group **Pen Options**

Element	Description
Default/ Thin/ Thick Line	If the parameters of drawing a line of a style to be created must be the same as those set for some system line (normal, thin or thick), activate the respective option. After activation of one of these options, the fields for entering width become unavailable. In changing the properties of a normal, thin or thick system line* the appearance of lines of a custom style, which was customized with a respective option, activate changes, too.
Thickness On Paper	Sets the line thickness for printing on paper (in millimeters).
Thickness On Screen	Line thickness for display (in pixels).

* System line customization is made in the dialog activated by the **Service — Settings... — System—Graphic editor— System lines** command.

Tab. 10.1.8. Elements of a group **Description of Dashed Curve**

Element	Description
Dash-Gap	A list of specified combinations of dash lengths and gap lengths. The line can be composed of a sequence of such combinations. To edit the dimensions of a dash and a gap in a combination, select this combination in a list and change the values in the fields Dash Length and Gap Length .
Dash Length, Gap length	Dimensions of dashes and gaps in millimeters.
Add	A button making it possible to create a new dash-and-gap combination. Dimensions of a dash and a gap in a new combination are the same as those in the combination selected before a new combination was created. A new combination is added to the end of a list.
Delete	A button making it possible to delete a dash-and-gap combination selected in a list.
Fragment	This button is used to add an KOMPAS-3D arbitrary fragment to a dash-and-gap combination. For more information, see Section 10.1.2.1.1.
X,Y	The fields for entering the values of the fragment origin offsets relating to the dash beginning.

Tab. 10.1.8. Elements of a group **Description of Dashed Curve**

Element	Description
Ignore the Styles of Fragment Lines	<p>If this option is enabled, then all lines of all fragments use width and color specified for a line style to be customized (using the button Color and the group Pen Options)</p> <p>If this option is disabled, the width and the color of fragment lines depend on line styles used in them, as follows:</p> <ul style="list-style-type: none"> ▼ Custom styles of a fragment line are replaced with a system line style <i>Normal</i> with current settings, i.e., screen and print color and width settings set in the system at the moment of including the fragment into a line style. ▼ System styles of a fragment line are saved in a line style with current properties (if system lines had other properties when a fragment was created, these properties are replaced with the current ones)*. Intermittent lines, such as <i>Dashed</i>, <i>Axial</i> etc., are replaced with solid lines.
Curve Always Terminates with Dash	<p>Activation of this option means that a line will start and end with a dash (not with a gap). The fitting is performed by proportional change of dash lengths.</p> <p>The status of this option is taken into account only if the dashes have zero length.</p>

* Properties of fragment lines are saved in a line style. They cannot be changed neither during, nor after style creation.

10.1.2.1.1. Line Style with Fragments

KOMPAS-3D makes it possible to obtain not only solid and dashed lines, but also the lines that include fragments, i.e. images that repeat throughout the entire line (Fig.10.1.11).

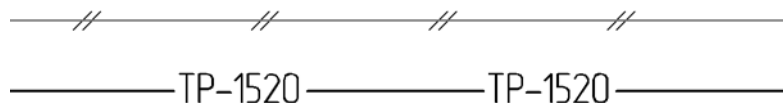


Fig. 10.1.11. Examples of lines containing fragments

The fragments are selected when a line style is customized — in the **Description of Dashed Curve** group of the Line Style Creation and Edition dialog (see Fig.10.1.10 on p. 1920). The fragments can be included in an dashed curve only.



To imitate a solid line, set a zero gap length.

To add a fragment in a dash-and-gap combination, select this combination in a list and click the **Fragment...** button.

A standard File Selection dialog will appear on the screen. In this dialog you must select a fragment **.frw* to be included in the line style (naturally, it must be prepared beforehand).

Then you must select the image location relative to a dash beginning. To do it, in the **X** field enter the value of the offset in dash direction (along a line), and in the **Y** field — the value of the offset which is perpendicular to the dash direction (perpendicular to a line).

The fragments are included in a line style in accordance with the following rules.

- ▼ One fragment can be included in one dash-and-gap combination.
- ▼ The origin of a fragment is offset by the specified values X and Y relative to a dash beginning in a dash-and-gap combination, and a fragment is not rotated and scaled.
- ▼ The origin of a fragment must fall within an interval dash length + gap length, i.e., the offset of a fragment along must be less than a total length of a dash-and-gap combination.
- ▼ The fragment included in a line style loses its connection with its source file.

To delete a fragment from a style, you must delete a dash-and-gap combination containing this fragment.

- ▼ Only curves, fills and texts having a True Type font are sent from a fragment to the contents of a “picture” in a line style. In this case letters transform into fills.
- ▼ Inherent colors of fills and texts contained in fragments are ignored. The color selected for dashes of the customized line style is used for them.
- ▼ The width and color of the image lines depend on the status of the **Ignore fragment line styles** option (see Table 10.1.8), may be the same as hatch width and colors in the line style that is being set up. They can also depend on the line styles used in fragments.

The lines, the style of which includes fragments, are drawn in a document according to the following rules.

- ▼ The fragments, the outline dimensions of which transcend the curve limits, are not drawn.
- ▼ The fragments are arranged in such a way that the X axis of each of them would be directed tangential to a curve in a dash beginning point.
- ▼ Hatches, fills and equidistant lines of the contours formed by the lines which include fragments, are constructed ignoring the fragments.

10.1.2.2. Practice of Creating Line Styles

This Section contains exercises enabling to master the technique of creating line styles (including the styles containing fragments).

Exercise 1. Creating a line style library

Task. Create a file of a line style library `user_lines.lcs`.

1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Line Styles...** A Style Set and Style Library Operation dialog will appear on the screen.
2. Click the right preview window. It will become active: a marquee will appear around it.
3. Click button **Show Library**.



4. The Open File dialog will appear on the screen. By default, the current folder is a folder \Sys, containing a system line style library *Graphic.lcs*. Select another folder, if necessary.
5. Enter the name of a new library, *user_lines*, in the **File Name** field.
6. Click the **Open** button. A dialog with a prompt for creating a file *user_lines.lcs* will appear on the screen.
7. Click the **Yes** button.

A new style library— *user_lines.lcs* will be created in the current folder.

The dialog with a prompt and the File Opening dialog will close, and a screen will display a Style Set and Style Library Operation dialog. The header of an active (right) window will contain a word “Library” and a full path to the Library created. The active preview window is empty, as it contains no styles yet.

Without closing the Style Set and Style Library Operation dialog, proceed to performing the next exercise.

Exercise 2. Creating a line style without fragments

Task. Create a line style shown in Fig. 10.1.12. Dimensions of dashes and gaps are shown in Fig. 10.1.13. Line width – 1.2 mm, color – dark blue.



Fig. 10.1.12. A sample of a line style



Fig. 10.1.13. Dimensions of dashes and gaps



1. Click the **Create** button in the Style Set and Style Library Operation dialog. A Line Style Creation dialog will appear on the screen.
 2. Enter value *100* into field **Number**.
 3. Enter the name of a style to be created, *Dashed Special*, in the **Name** field.
 4. In the **Curve Type** group select the **Dashed** option.
 5. Activate the **Serves as Hatch Border** option, so that the line of a style to be created could border the hatch.
 6. Select the **Assignable** option in the **Pen options** group.
 7. Enter *1.2* in the **Width on Paper** field, and *2* in the **Width on Screen** field.
 8. Click the **Color** button and select a dark blue color for a line.
- The style created consists of four dash-and-gap combinations (Fig.10.1.14).

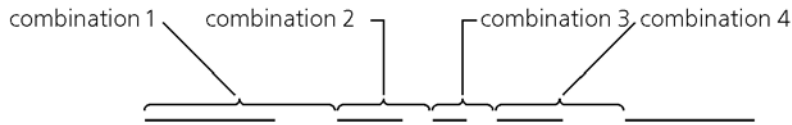


Fig. 10.1.14. Dash-and-gap combinations in a Dashed special line style.

List **Dash-Gap** in the **Description of Dashed Curve** group while it contains one dash-and-gap combination. By default, the dash length is equal to the gap length and equal to 1mm.

- Change these values for the following ones: dash length— 4, gap length— 2.

The changes made will be sent to a **Dash-Gap** list of combinations. The **Preview** field will display a line consisting of 4-mm dashes separated by 2-mm gaps.

- Click the **Add** button.

A new combination duplicating the existing one will appear in a **Dash-Gap** list.

- Make a dash length to be equal to 2, and that of a gap — to 1.

This change will also be fixed in a list of combinations and in the **Preview** field.

- Add the third and the fourth combinations in accordance with Fig. 10.1.14.

- Activate the **Line Always Terminates with Dash** option, so that the line of a style under creation will begin and end with dashes, not with gaps.

Customizing of a new style is completed (Fig.10.1.15).

Click the **OK** button in a Style Creation dialog.

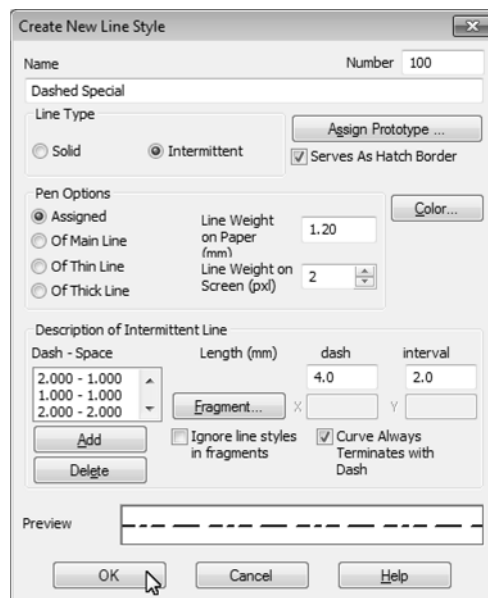


Fig. 10.1.15. Dashed Special Line style settings

The Style Creation dialog will be closed. A Style Set and Style Library Operation dialog will remain on the screen. The name of the first style of the current library — the style created by you — *Dashed Special* — will appear in the right preview window.

14. Use of Style Library Sets dialog.

Exercise 3. Check of a line style created.

Task. Check the compliance of the **Dashed Special** line style created by you with the requirements set forth in the task for the Exercise 2.

1. Create a KOMPAS-3D graphic document.
2. Apply the **Tools — Geometry — Lines — Line** command.
3. Click in the **Style** field on the property bar.
4. Select the **Another Style** option in the list that expanded.
5. In the Current Style Selection dialog that appears on the screen, click the **Style Manager...** button.
6. Click **Add styles** in the Style Manager window that appears.



7. Click the **Show Library** button in the right-hand area of the Manager window.
8. Select *user_lines.lcs* file in the file opening dialog that appears and click **Open**. The style list is displayed in the left part of the Line Style Manager. In this case, the list has exactly one line, which is highlighted.

To be able to use a style from a library in a document, you must add or embed it into the document. The **Added** style maintains a relationship with the library, which means that the changes of this style made in the library will affect the document; the **embedded** style bears no relationship with the library and in fact does not differ in any way from a style created directly in the document.



9. Click **Add library style** or **Embed Style into Document** button+.



10. Close the Line Style Manager by clicking **OK**.

The current style selection dialog will appear.

11. If the style was added, go to the **Library** tab in the current style selection dialog, and if the style was embedded, go to the **Embedded** tab.
12. Select the only style, *Dashed Special*, in the list of the styles on the tab and click the **Select** button.

The dialog will close and the selected style will become the current style: It will be displayed in the **Style** field.

13. Create a line, 50mm long.

The resulting line must have a dark blue color and a screen width of 2 pixels.



Since the **Line Always Terminates with Dash** option was activated when a line style was customized, the fitting of dashes is performed when a line is drawn — such proportional change of their lengths which is made in order to meet the above-mentioned condition. Due to the fact that a line length by far exceeds dash lengths, their deviations are not visible. In case of a small line length, the alterations of dash lengths may become visible. In case the dash length is less than the aggregate of lengths of all dash-and-gap combinations specified in the process of customizing a line style, a solid line is drawn. If the **Line Always Terminates with Dash** option is deactivated, the fitting of dashes is not performed.

14. To check whether the width set for printing (1.2mm) is met, print the document.
15. To check whether a created line is a hatch border, construct a circle such that the created line would be its chord.
16. Create a hatch, indicating its base point inside the circle. The hatch must appear only in that part of the circle that is delimited by the line segment.

Exercise 4. Creating a line style with fragments

Task. Create a line style **Oil-proof Capping** (Fig. 10.1.16,10.1.17). The width and color of a line correspond to a normal line.

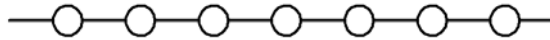


Fig. 10.1.16. A sample of a line style

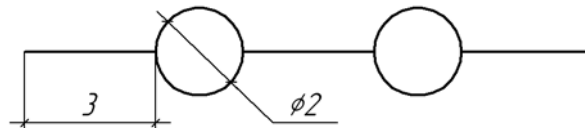


Fig. 10.1.17. Dimensions of elements

1. Create a fragment, which will be included in a line style.
 - 1.1. Create a new fragment.
 - 1.2. In this fragment create a circle with the radius of 1mm with its center point at the coordinate origin. A circle can have any line style.
 - 1.3. Save the fragment as *line_style.frw*.
2. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Line Styles...**
3. Open the *user_lines.lcs* library, created when you did the Exercise 1.
 - 3.1. Click button **Show Library**.
 - 3.2. Select the *user_lines.lcs* file in the File Opening dialog which appears on the screen.

A full library name shall appear in the header of the active preview window; and in the window itself — a list of styles contained in this library.
4. Create a new style.





- 4.1. Click the **New** button.
A Line Style Creation dialog will appear on the screen.
- 4.2. Enter value *101* into field **Number**.
- 4.3. Enter the name of a style to be created— **Oil-proof Capping** — in the *Name* field.
- 4.4. In the **Curve Type** group select the **Dashed** option.
- 4.5. Activate the **Serves As Hatch Border** option.
- 4.6. Select an **Normal Line** option in the **Pen Options** group.
- 4.7. **Dash-Gap** list in the **Description of Intermittent Line** group contains one dash-and-gap combination. By default, the dash length is equal to the gap length and equal to 1mm. Change these values for the following ones: dash length— 3, gap length— 2.



If before clicking the **Create** button, a *Dashed Special* style was selected in the dialog, this style is used by default as a prototype. Hence, the **Dash-Gap** list will contain four combinations. Delete three of them using the **Delete** button. Edit the remaining combination by changing dash-and-gap lengths.

The changes made will be sent to a **Dash-Gap** list of combinations. A line consisting of 3-mm dashes separated by 2-mm gaps will be displayed in the **Preview** field.

- 4.8. Activate the **Line Always Terminates with Dash** option so that the line of a style created will begin and end with dashes.
- 4.9. Click the **Fragment...** button. The Open File dialog will appear on the screen.
- 4.10. In this dialog, select the file created when p.1 was performed and click the **Open** button.
The selected fragment will be added to the current dash-and-gap combination (selected in a list). It will be displayed in the **Preview** field: an image of a selected fragment— a circle — appears in it.

Since the values in the **X** and **Y** fields— fragment origin offsets relative to a dash beginning— are by default equal to zero, the dashes overlap the circles.

To ensure the compliance of the style to be created with a sample, you must locate a circle center point in the middle of a gap (Fig. 10.1.18).

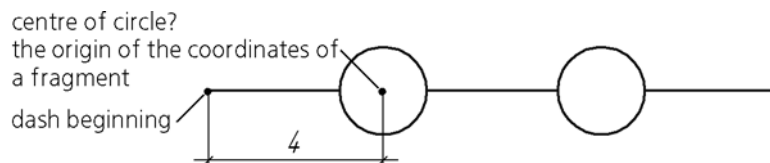


Fig. 10.1.18. Location of a circle in relation to a dash beginning.

- 4.11. Enter value **4** into the *X* field.
The circles in the **Preview** field will be moved by 4mm to the right, thereby occupying the required place.
Vertical offset is not required, therefore value **0** in the *Y* field should remain unchanged.

- 4.12. Customizing of a new style is completed (Fig.10.1.19). Click the **OK** button in the Line Style Creation dialog.

The Style Creation dialog will close and the name of the second style— *Oil-proof Capping* will appear in the active preview window.

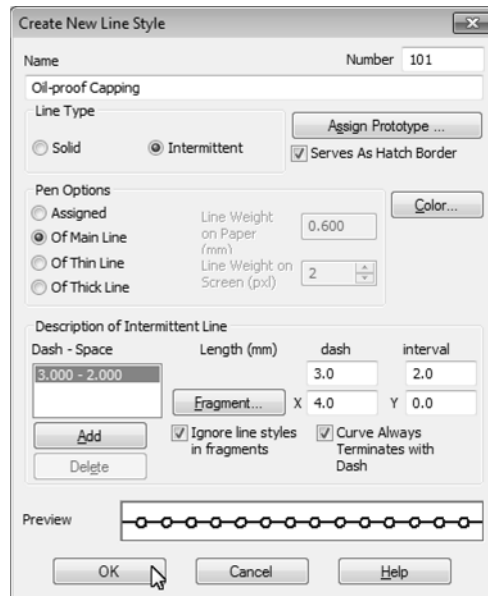


Fig. 10.1.19. Settings of line style Oil-proof Capping

5. Close the Style Set and Style Library Operation dialog by clicking the **Exit** button.
6. Make sure that the style is created correctly by performing the operations similar to those described in the Exercise 3.

Exercise 5. Creating a line style with fragments Independent work

Task. Create a line style **Oil-proof Capping** (Fig. 10.1.20, 10.1.21). The width and color of a line correspond to a normal line.

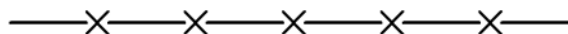


Fig. 10.1.20. A sample of a line style

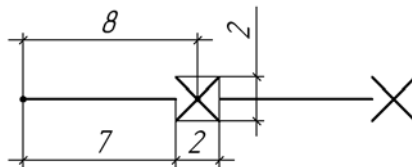


Fig. 10.1.21. Dimensions of elements

10.1.3. Custom Hatch Style

To create a hatch style, perform the following operations.

1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Hatch Styles...**



A Style Set and Style Library Operation dialog will appear on the screen.

2. Select a new style storage location.



To achieve this, use the **Show Library...**, **Show Set...**, or **Show Document** buttons. Style storage locations are described in Section 10.1.1.1 on p. 1907.



3. Click the **New** button.

The dialog for creating and editing hatch style will appear on the screen.



4. Customize the parameters of a new style (see Section 10.1.3.1) and close the dialog.

The name of the style created will appear in an active preview window of the Style Set and Style Library Operation dialog.



After reviewing the material set forth in Section 10.1.3.1, it is recommended to do the exercises of Section 10.1.3.2. This will allow you to practice the described technique of creating hatch styles.

10.1.3.1. Customizing a Hatch Style

Customizing a hatch style— description of **groups of recurrent lines** component. In this Section and in Section 10.1.3.2 the word “line group” shall mean a group of hatch lines.

A plain hatch (for example, hatch of a system style *Metal*) consists of one group, and a composite hatch (for example, *Non-metal*) consists of several groups overlapping each other.

A hatch style is customized in a Style Creation and Edition dialog (Fig.10.1.22). The control elements of this dialog are represented in Table 10.1.9.

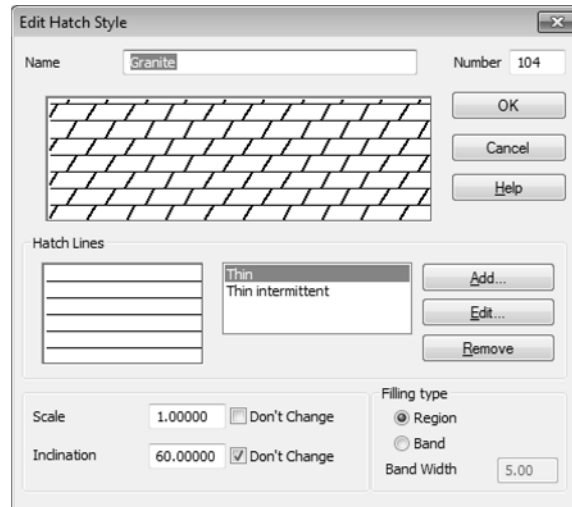


Fig. 10.1.22. Dialog to Create or Edit Hatch Style

Tab. 10.1.9. Dialog to Create or Edit Hatch Style

Element	Description
Name	A field for entering (editing) a style name. It is recommended to name styles using informative names that accurately represent their purpose. In the future this will simplify the search for a style in a list.
Number	A field for entering (editing) a style number. The rules of style numbering are set forth in Section 10.1.1.2.4 on p. 1916.
Preview Window	Contains the image of a current hatch view. Any change of the properties will be immediately displayed in this window.
Lines for hatch	A group of elements for operation with lines forming a hatch. This group contains a list of line groups, a preview window for the selected group, and the control buttons for the groups. The Add... button makes it possible to include a new group of lines into a style. After clicking it, a Hatch Line Properties Customizing dialog appears on the screen (see Table 10.1.11). A new group is always added at the end of the list. The Edit... button makes it possible to change the properties of the group selected in a list. After clicking it, a Hatch Line Properties Customizing dialog likewise appears on the screen. The Delete button makes it possible to exclude the group, selected in a list, from a hatch.

Tab. 10.1.9. Dialog to Create or Edit Hatch Style

Element	Description
Scale	This field serves for entering the image scale [*] . If you want to be able to change the hatch scale when a hatch of this style is created, deactivate the Do not change ^{**} option.
Slope Angle	The field for the slope of a hatch image. If it is required to make it possible to change the hatch slope when a hatch of a given style is created, deactivate the Do not change option ^{**} .
Filling type	Area option makes it possible to set a solid hatch fill within selected borders. Band option makes it possible to set a hatch in the form of a band along a border. The Band Width field makes it possible to set a width (in millimeters) of a band, which will be filled with a hatch.

* The following values are to be multiplied by the factor set in this field: **X**, **Y**, **deltaX**, **deltaY** (see Table 10.1.10 on p. 1934), as well as hatch and gap lengths in intermittent lines. The line width remains unchanged.

** In this case, when a hatch is created in a graphic document, a corresponding field appears on the properties bar. Inclination

10.1.3.1.1. Setting Hatch Line Properties

When a hatch style is created, the relative position of groups is defined by the values of their offset and rotation relative to the origin of some arbitrarily selected Cartesian coordinate system (Fig. 10.1.23).



The view of a hatch itself does not depend on the location of a coordinate system when a hatch style is created. Therefore, the coordinate system location should be selected on the basis of operational convenience. You may practice selecting the location of a coordinate system, performing Exercises 7 and 9 from Section 10.1.3.2.

When a hatch is created in a graphic document, the hatch is located in such a manner that the origin of its coordinate system would coincide with the origin of coordinate system of a fragment or a drawing view. The rotation angle of a hatch coordinate system is defined by a value specified in the **Slope** field of a Hatch Creation and Edition dialog (see Table 10.1.9).

Parameters characterizing the location of a group of lines in a hatch coordinate system are represented in Table 10.1.10.

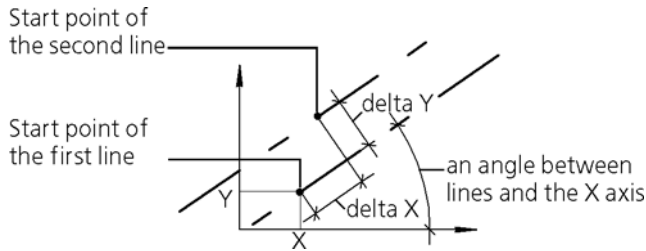


Fig. 10.1.23. Properties of a hatch line group

Tab. 10.1.10. Properties of a hatch line group

Parameter	Description
angle	The lines group slope angle to the X axis of the hatch coordinate system.
X	Abscissa of a start point of the first line *.
Y	The ordinate of a start point of the first line.
deltaX	The subsequent line move relative to a preceding line (for groups of dashed lines).
deltaY	Spacing of lines (a distance between neighboring lines).

* The start point of a solid line is its arbitrary point. The start point of a dashed line is the beginning of a dash in the first dash-and-gap combination (see Table 10.1.8 on p. 1922).

Selection of lines forming a group and setting the location of a group in a hatch coordinate system is performed in the Line Properties Customizing dialog (Fig. 10.1.24).

The control elements of this dialog are described in Table 10.1.11.

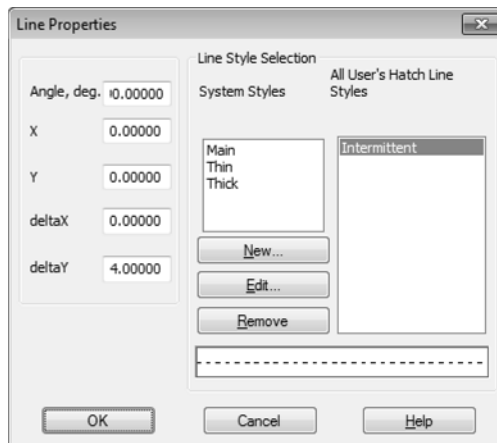


Fig. 10.1.24. Hatch Line Settings dialog

Tab. 10.1.11. Hatch Line Settings dialog

Element	Description
Selection Line Style	Group of elements (see Table 10.1.12 on p. 1935), which can be used to select the style of lines making up the group being set up.
Angle, X, Y, deltaX, deltaY	The fields for entering the values of parameters specifying the location of a group of lines in a hatch coordinate system (see Fig. 10.1.23). The line values are set in millimeters.

Tab. 10.1.12. Elements of the group **Line Style Selection**

Element	Description
System Lines	A list of system lines which can form a hatch line group to be customized.
All User's line styles of the current hatch	A list of custom lines which can form a hatch line group to be customized.
New...	The button making it possible to create a new custom line style. The button is accessible if the active element of a dialog is a list of custom lines. Activation is performed by a click in this list. After clicking the New... button the Line Style Creation and Edition dialog will appear on the screen (see Table 10.1.6 on p. 1921)*.
Edit...	A button which allows editing of a custom line style selected in a list. After clicking this button, the Line Style Creation and Edition dialog appears on the screen.
Delete	A button making it possible to delete a custom line style selected in a list. When deleting a user style, all lines of this hatch for which the style was used will be drawn by the system <i>Normal</i> line style.
Preview Window	Contains an image of a line selected in a list.

* When a line style for a hatch is created, the addition of fragments into dash-and-gap combinations is impossible, and therefore the corresponding elements are not available in the Line Style Creation dialog.

10.1.3.2. Practice in Creation of Hatch Styles

Hatches which meet GOST requirements are supplied as part of KOMPAS-3D system. You can create hatch styles meeting other standards — ISO, DIN and others.

This section contains exercises for mastering the technique of hatch style creation on the basis of the hatches samples, the use of which is required by German standard DIN 201.

**Exercise 6. Creation of a hatch style library.
Independent work**

Task. Create a hatch style library `user_hatches.lhs`.

1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Hatch Styles....**
2. Create a hatch style library `user_hatches.lhs` in a similar way as you created a line style library in Exercise 1.

Without closing the Hatch Style Set and Hatch Style Library Operation dialog, proceed to doing the next exercise.

Exercise 7. Creating a Porcelain hatch style

Task. Create a **Porcelain** hatch style (Fig. 10.1.25). Hatch line style— Thin.

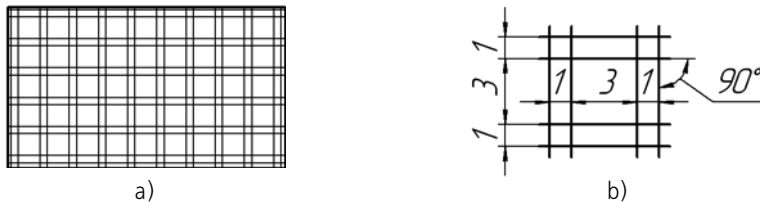


Fig. 10.1.25. Hatch style Porcelain:
a) hatch sample, b) location of hatch elements

This hatch consists of four groups of solid lines (Fig. 10.1.26).

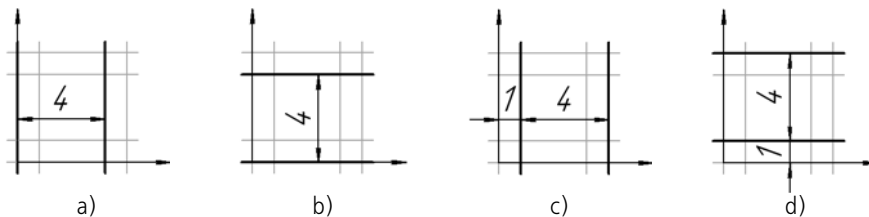


Fig. 10.1.26. Groups of lines forming a Porcelain hatch:
a) group 1, b) group 2, c) group 3, d) group 4

The location of a coordinate system is selected in such a way that it will be convenient to define the location of groups. The parameters characterizing this location are represented in Table 10.1.13.

Tab. 10.1.13. The parameters characterizing the location of groups of lines in a hatch coordinate system

Group	Angle	X	Y	deltaY
Group 1	90°	0	0	4

Tab. 10.1.13. The parameters characterizing the location of groups of lines in a hatch coordinate system

Group	Angle	X	Y	deltaY
Group 2	0°	0	0	4
Group 3	90°	1	0	4
Group 4	0°	0	1	4



1. Click the **Create** button in a Hatch Style Set and Hatch Style Library Operation dialog.



Make sure that the *user_hatches.lhs* library window will be active. Otherwise, a new style will be added to a library (a set or a document) opened in a neighboring window.

A Hatch Style Creation dialog will appear on the screen. By default, it contains settings corresponding to the system hatch style *Metal*.

2. Enter value *100* into field **Number**.
3. Enter the name of a style to be created — **Porcelain** — into the *Name* field.
4. Enter **1** into the *Scale* field. Do not activate the **Do not change** option, in order to enable the changing of its scale when creating a hatch of a given style in a drawing.
5. Enter **0** in the *Slope* field as a hatch must not be rotated relative to an absolute coordinate system of a drawing (a fragment). Activate the **Do not change** option, so as to make it impossible to change a hatch slope when creating a hatch in a drawing.
6. Since a hatch should be formed of thin lines, you can use an already existing group of lines. To set its parameters properly, click the **Edit...** button.

A Hatch Line Properties Customizing dialog will appear on the screen. You must set there the parameters characterizing the location of the first group of lines in a hatch coordinate system. Since the group should be formed of thin lines, the **Thin** style is selected in the **System Styles** list.

- 6.1. Enter **90** into the *Angle* field.
- 6.2. Enter **0** in the *X* field.
- 6.3. Enter **0** into the *Y* field.
- 6.4. Enter **4** in the *deltaY* field.
- 6.5. Here the customizing of group 1 has been completed. Click **OK**.

The hatch line parameters dialog is now closed. A Hatch Style Creation dialog will remain on the screen. In a small dialog preview window, the current group (selected in a list) of lines is displayed and hatch appearance is displayed in a large preview window. While a hatch contains only one group, the images in both windows will be the same.

7. Click the **Add...** button to add the second group of lines to a hatch.

A Line Properties Customizing dialog will appear on the screen.

- 7.1. Select the **Thin** style in the **Line Styles** group.
- 7.2. Set the parameters of the second group of lines in accordance with Table 10.1.13 and close the Line Properties Customizing dialog.

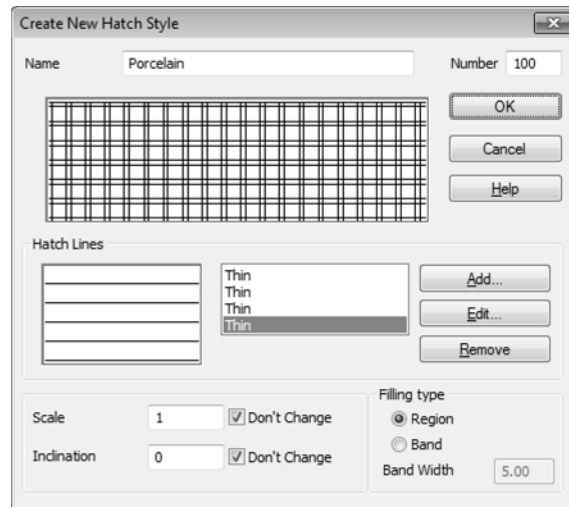


Fig. 10.1.27. Creating a Porcelain hatch style

8. Similarly, add the third and the fourth groups of lines in a hatch. Customizing the *Porcelain* hatch style is completed (Fig. 10.1.27).
9. Close the Hatch Style Creation dialog by clicking the **OK** button. A Hatch Style Set and Hatch Style Library Operation dialog will remain on the screen. The *Porcelain* hatch style created by you will appear in the *user_hatches.lhs* library.
10. Close the Style Set and Style Library Operation dialog by clicking the **Exit** button.

Exercise 8. Check of the hatch style created

Task. Check that the **Porcelain** hatch style created by you is customized properly.

1. Create a KOMPAS-3D graphic document.
2. Create a closed contour (a rectangle, a circle, etc.) in it, using the *Normal* system line style.
3. Apply the **Tools — Hatch** command.
4. Click in the **Style** field on the property bar.
5. Select the **Another Style** line in the displayed list.
6. In the Current Style Selection dialog, which appears on the screen, click the **Library...** button.
7. In the File Opening dialog, which appears on the screen, select a library created by you *user_hatches.lhs*.
8. On the **Library** page of the Current Style Selection dialog select the library style name — *Porcelain* — and click the **Select** button. The dialog will close and the selected hatch style will become a current one. It will be displayed in the **Style** field.
9. Make sure that the **Step** field, allowing the change of a hatch scale, is available on the properties bar, and the **Slope** field is not available.
10. Click the **Create Object** button on the Advanced Control Panel.

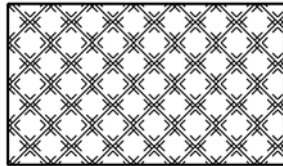
11. Look at the hatch created. Make sure that a hatch pattern is the same as that shown in Fig.10.1.25, and the used line style is *Thin*.



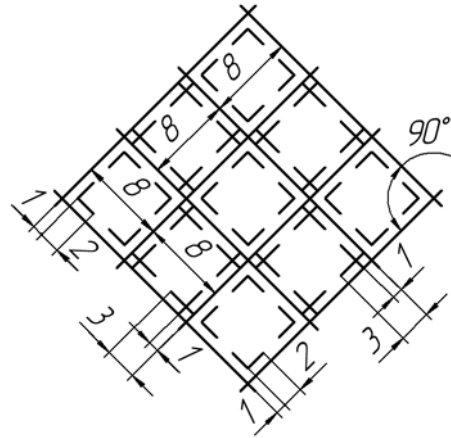
If, creating a hatch, you selected a scale different from 1 on the properties bar, the distance between the lines will change in accordance with a specified factor.

Exercise 9. Creation of a Thermoplast hatch style

Task. Create a **Thermoplast** hatch style (Fig. 10.1.28). Hatch line style— Thin.



a)



b)

Fig. 10.1.28. Thermoplast hatch style:
a) sample of a hatch, b) size and location of hatch elements

This hatch consists of two groups of solid lines and four groups of dashed lines (Fig. 10.1.29). A hatch coordinate system is a system, the X-axis of which coincides with one of the solid lines located at an angle of 45° from horizontal, and the Y-axis of which coincides with a solid line at an angle of 135° . In Fig.10.1.29 the hatch is rotated in such a way that the abscissa axis of its coordinate system would be located horizontally.

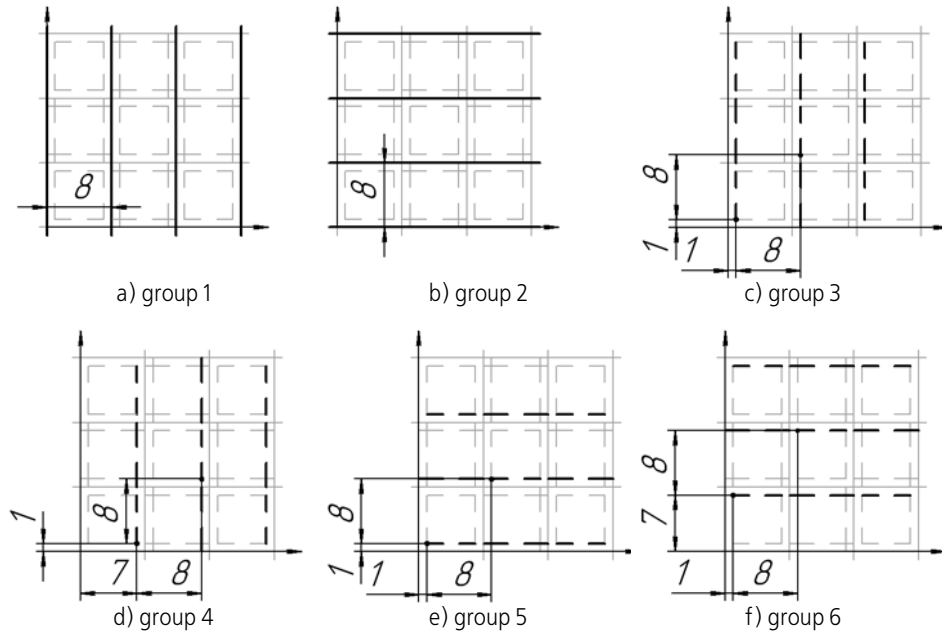


Fig. 10.1.29. Groups of lines forming a Thermoplast hatch:
a) group 1, b) group 2, c) group 3, d) group 4, e) group 5, f) group 6

Parameters characterizing the location of groups of lines in a hatch coordinate system are shown in Table 10.1.14.

Tab. 10.1.14. The parameters characterizing the location of groups of lines in a hatch coordinate system

Group	Angle	X	Y	deltaX	deltaY
Group 1	90°	0	0		8
Group 2	0°	0	0		8
Group 3	90°	1	1	8	8
Group 4	90°	7	1	8	8
Group 5	0°	1	1	8	8
Group 6	0°	1	7	8	8

Groups 3–6 contain dashed lines. All of them are formed of the same dash-and-gap combinations. These combinations and dash-and-line lengths are shown in Fig. 10.1.30.

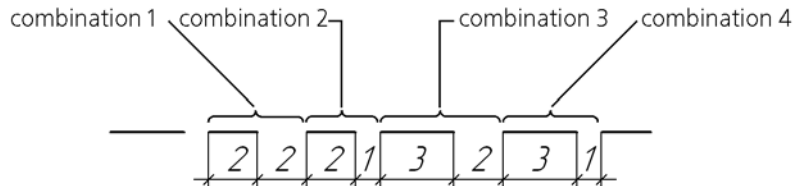


Fig. 10.1.30. Dash-and-gap combinations in dashed lines of Thermoplast hatch

1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Hatch Styles...**

The Hatch Style Set and Hatch Style Library Operation dialog will appear on the screen.

2. Open the *user_hatches.lhs* library created when you did Exercise 1.



- 2.1. Click button **Show Library**.

- 2.2. Select the *user_hatches.lhs* file in the File Opening dialog, which appears on the screen.

A full library name shall appear in the header of the active preview window; and in the window itself — a list of styles contained in this library.



3. Click the **New** button.



Make sure that the *user_hatches.lhs* library window will be active. Otherwise, a new style will be added to a library (a set or a document) opened in a neighboring window.

A Hatch Style Creation dialog will appear on the screen. If, before the **Create** button is clicked, the existing *Porcelain* style was selected in the list of styles of the current library, its parameters are contained in the Style Creation dialog (i.e., the selected style is used as a prototype). In this case, using the **Delete** button, delete all lines except for the first one (**Thin**) from the list.

If the *Porcelain* style was not selected, the dialog contains the parameters corresponding to the *Metal* hatch system style.

4. Enter value *101* into field **Number**.
5. Enter the name of the style to be created — **Thermoplast**, in the *Name* field.
6. Enter **1** into the *Scale* field. Do not activate the **Do not change** option, in order to enable the changing of its scale when creating a hatch of a given style in a drawing.
7. Enter **45** in the *Slope* field as the hatch coordinate system is rotated in relation to the absolute coordinate system of a drawing (a fragment) by 45°. Activate the **Do not change** option, so as to make it impossible to change a hatch slope when creating a hatch in a document.
8. Having completed steps described in paragraphs 6–7 of Exercise 7, edit the existing group of thin lines and add the new one. Parameters characterizing the location of these groups in a hatch coordinate system are represented in Table 10.1.14. To monitor the correctness of your operations, compare the dialog that you see on the screen with the dialog shown in Fig.10.1.31.

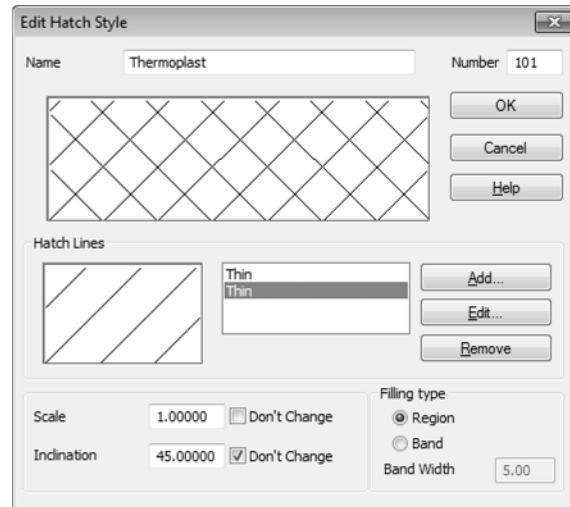


Fig. 10.1.31. Creating the first and the second groups of lines

9. Click the **Add...** button to add the third group of lines to a hatch.
A Line Properties Customizing dialog will appear on the screen.
The third group contains dashed lines. Since all available system lines are solid, you must create a custom line style.
 - 9.1. Click on the **All User Hatch Line Styles** window.
 - 9.2. Click the **New...** button which has become accessible.
A Line Style Creation dialog will appear on the screen.

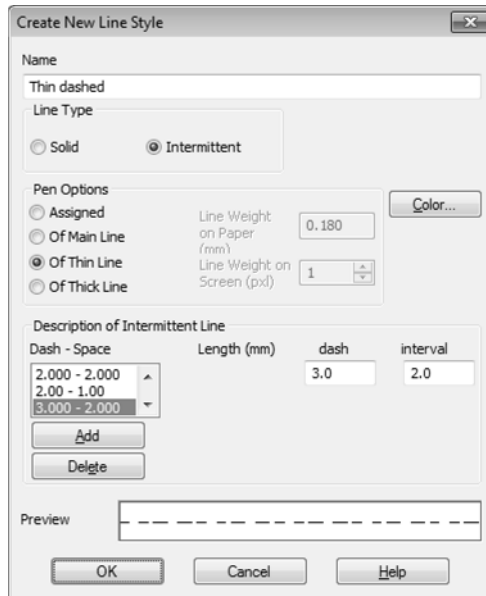


Fig. 10.1.32. Customizing a User's style

- 9.3. Enter **Thin Dashed** in the *Name* field.
- 9.4. In the **Curve Type** group select the **Dashed** option.
- 9.5. Select the **Thin Line** option in the **Pen Options** group.
- 9.6. Click the **Color...** button and set black color for the line.
- 9.7. The line consists of dash-and-gap combinations (see Fig.10.1.30 on p. 1941). The **Dash-Gap** list contains by default one combination, the dash-and-gap lengths in which are the same and equal to 1 mm. Enter the value 2 in the dash length field and the gap length field. This change will be sent to the **Dash-Gap** list and also be displayed in a preview window.
- 9.8. Click the **Add** button. The second dash-and-gap combination will appear in the list. Edit it according to Fig.10.1.30.
- 9.9. Similarly, add the third and the fourth combinations.
- 9.10. Customizing of the User's style is completed (Fig. 10.1.32). Click the **Exit** button in the dialog.
Line Style Creation dialog will close. A Line Properties Customizing dialog will remain on the screen.
10. The **All User Hatch Line Styles** list contains the name of a line style created by you — **Thin Dashed**. Select this style and enter the values characterizing the location of the third group of lines in a hatch coordinate system (see Table 10.1.14 on p. 1940).
11. Customizing the third group of lines is completed. Close the Line Properties Customizing dialog by clicking the **OK** button.

A Hatch Style Creation dialog will remain on the screen. A **Thin Dashed** line will appear in the list of lines, and an image of the respective group of lines will appear in the hatch preview window.

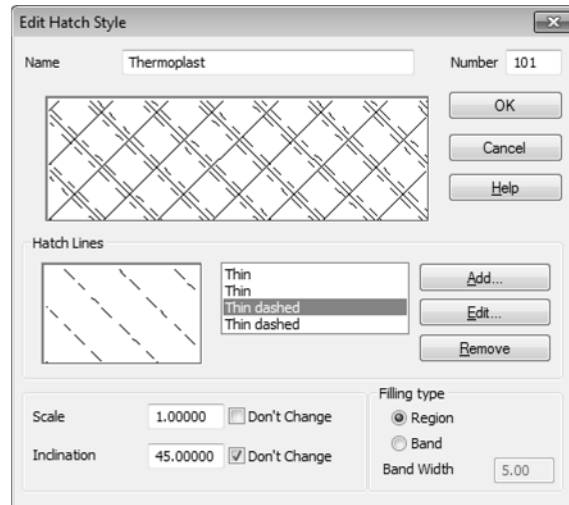


Fig. 10.1.33. Creating the third and the fourth groups of lines

12. Click button **Add...**
13. In the Line Properties Customizing dialog, which appears on the screen, select the **Thin Dashed** style and set the location parameters for the fourth group of lines. Then close the dialog by clicking the **OK** button. To monitor the correctness of your operations, compare the dialog that you see on the screen with the dialog shown in Fig.10.1.33.
14. Similarly, add the fifth and the sixth group of lines to a hatch.
Here, creating a *Thermoplast* hatch style is completed (Fig. 10.1.34).

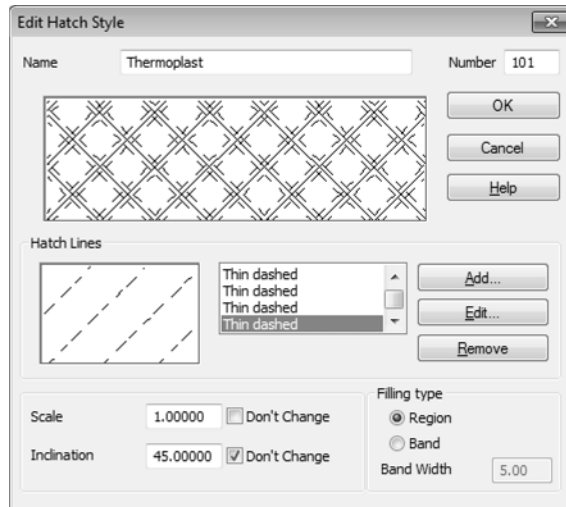


Fig. 10.1.34. Creating the fifth and the sixth groups of lines

15. Close the Style Creation dialog by clicking the **OK** button.
A Hatch Style Set and Hatch Style Library Operation dialog will remain on the screen. Now the current *user_hatches.lhs* library contains two styles: *Porcelain* and *Thermoplast*.
16. Close the dialog by clicking the **Exit** button.
17. Make sure that the style is created correctly by performing the operations similar to those described in the Exercise 8 on p. 1938.

Exercise 10. Creating hatch styles. Independent work

Task. Create a hatch style **Insulating Layer** (Fig.10.1.35). Hatch line style— Thin.

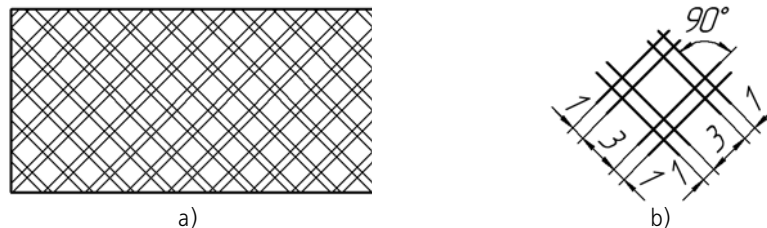


Fig. 10.1.35. Insulating Layer hatch style:
a) sample of a hatch, b) size and location of hatch elements

Task. Create a hatch style **Salts** (Fig. 10.1.36). Hatch line style— Thin.

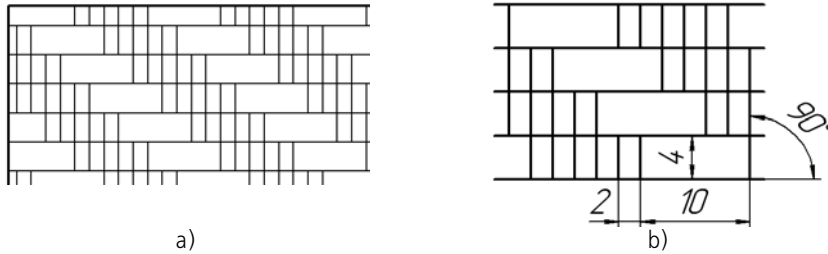


Fig. 10.1.36. Salts hatch style:
a) sample of a hatch, b) size and location of hatch elements

Task. Create a hatch style **Granite** (Fig. 10.1.37). Hatch line style— Thin.

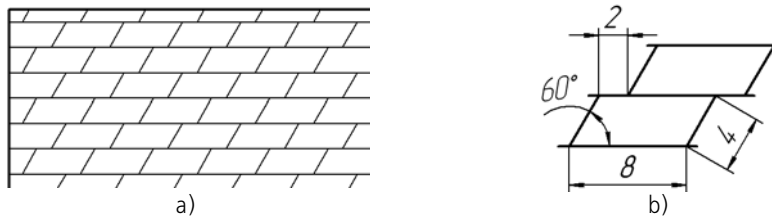


Fig. 10.1.37. Granite hatch style
a) sample of a hatch, b) size and location of hatch elements

Task. Create a hatch style **Resin** (Fig.10.1.38). Hatch line style— Thin.

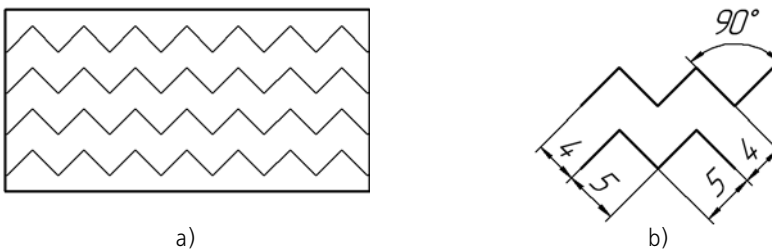


Fig. 10.1.38. Resin hatch style:
a) sample of a hatch, b) size and location of hatch elements

10.1.4. Custom Text Style

To create a text style, perform the following operations.

1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Text Styles...**
A Style Set and Style Library Operation dialog will appear on the screen.
2. Select a new style storage location.



To achieve this, use the **Show Library...**, **Show Set...**, or **Show Document** buttons. Style storage locations are described in Section 10.1.1.1 on p. 1907.



3. Click the **New** button.

The dialog for creating and editing text style will appear on the screen.



4. Customize the parameters of a new style (see Section 10.1.2.1) and close the dialog.

The name of the style created will appear in an active preview window of the Style Set and Style Library Operation dialog.



After reviewing the material described in Section 10.1.4.1, it is recommended to do exercises 11–13 from Section 10.1.4.2. It will allow you to practice the described technique of creating text styles.

10.1.4.1. Customizing a Text Style

A text style is customized in the Style Creation and Edition dialog (Fig. 10.1.39). The control elements of this dialog are represented in Table 10.1.15.

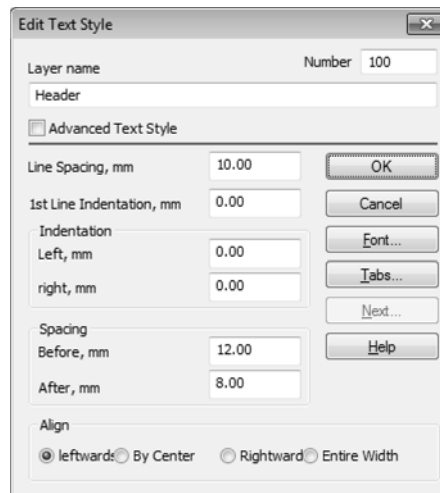


Fig. 10.1.39. Text Style Creation and Editing dialog

Tab. 10.1.15. Text Style Creation and Editing dialog

Element	Description
Name	A field for entering (editing) a style name. It is recommended that informative style names be given which fairly represent the function of such styles. In the future this will simplify the search for a style in a list.
Number	A field for entering (editing) a style number. The rules of style numbering are set forth in Section 10.1.1.2.4 on p. 1916.

Tab. 10.1.15. Text Style Creation and Editing dialog

Element	Description
Advanced Text Style	Option allowing the setting of additional parameters of a text style. After activating it, the Next... button, which activates an Advanced Text Style Customizing dialog, becomes available.
Line Spacing	The distance between the base text lines in a paragraph (see Fig. 10.1.40).
1st Line Indent	Indent to the right in the first line of a paragraph (see Fig. 10.1.40) *.
Offset at left Offset at right	Distances between the left and right border of the input field and the corresponding text borders (see Fig. 10.1.40) *.
Spacing	The distance between two contiguous paragraphs. It is formed from two values— an interval after the preceding paragraph and the interval before the following paragraph.
Alignment	A group of switches that allows to set the method to align paragraphs (to the left border , to the right border , centering , and alignment by two borders).
Font	The button allowing customizing of font options (see Section 10.1.4.1.1).
Tabulation	The button allowing customizing tabulation options (see Section 10.1.4.1.2).
Next...	The button allowing customizing an advanced text style (see Section 10.1.4.1.3).

* Note that due to the presence of spaces in font characters, which are necessary to prevent the “sticking” of letters to each other, this distance may slightly differ from that set in the dialog.

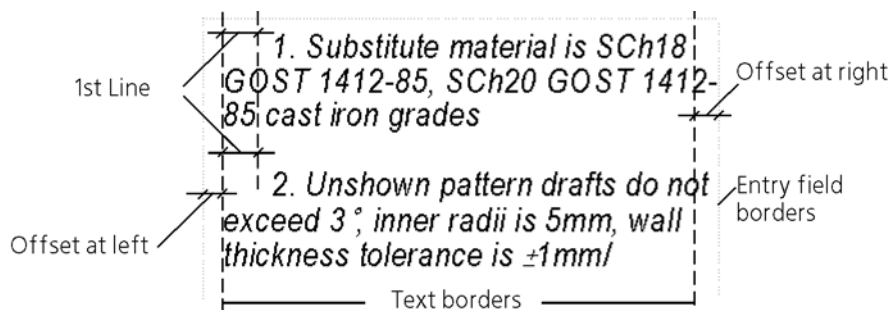


Fig. 10.1.40. Paragraph Properties

10.1.4.1.1. Font

Font options are customized in a dialog (Fig. 10.1.41) which appears after clicking the **Font...** button of the Text Style Creation and Edition dialog.

Control elements of the Font Options Customizing dialog are represented in Table 10.1.16.

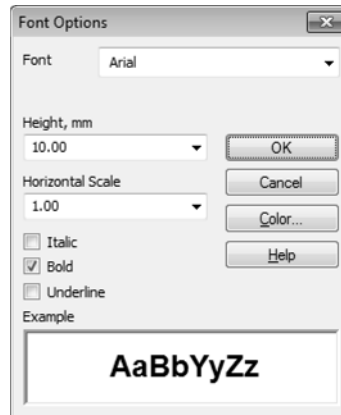


Fig. 10.1.41. Font options setup dialog

Tab. 10.1.16. Font options setup dialog

Element	Description
Font	A list allowing selection of any font set in Windows except for bitmap fonts.
Height	A field for entry or selection of font size. Note, that the size of the standard drawing font is the vertical size of a capital letter, expressed in mm.
Compression	A field for entry or selection of font spacing. If spacing is less than 1, this means to decrease of the width of letters and spacing between them by the specified factor, and if more than 1, to increase.
Color	A button for selecting the color of text characters.
Italic	An option allowing the setting of an italics style of characters.
Bold	An option allowing the setting of a thickened (bold) style of characters.
Underlined	An option allowing the setting of character underlining.
Example	A preview window for a font with current options.

10.1.4.1.2. Tabbing

Tabbing is customized in a dialog (Fig. 10.1.42) which appears on the screen after clicking the **Tabs...** button of the Text Style Creation and Edition dialog.

Control elements of Tabbing Customization dialog are represented in Table 10.1.17.

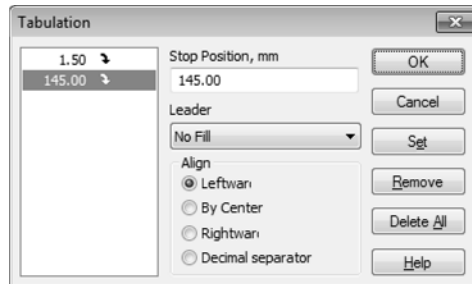


Fig. 10.1.42. Tabulation Settings Dialog

Tab. 10.1.17. Tabulation setup dialog

Element	Description
Ref.No.	Sets the tab indents in millimeters.
Leader	A list permitting one to select a method of entering tabbing options.
Alignment	A group of switches that manages the method to align text relative to the position of tabulation (Table 10.1.18). Text alignment by tabbing can be used, for example, for constructing a table of contents.
Set	A button allowing addition of the adjustable tab in a list.
Delete	A button allowing deletion of the selected tab.
Delete all	A button making it possible to clear at once the entire list of tabs.

Tab. 10.1.18. Methods to align text relative to the position of tabulation

Method	Description	Example
Left	A part of a line entered after a tab is located to the right from the tab position. Several lines beginning from this tab will be left-aligned.	Columnar layout. Each paragraph consists of one line and contains the same number of tabs and columns.
Centered	A line part entered after a tab is positioned symmetrically about the tab position.	Headers. Tabs are entered in each line before the header text.

Tab. 10.1.18. Methods to align text relative to the position of tabulation

Method	Description	Example
Right	A line part entered after a tab is positioned to the left from the tab position, i.e., this part of a line ends in a tab position. Several lines containing such tabs will be right-aligned.	Contents. A right tab with fill characters is entered between a header text and a page number.
Decimal point	A point that is entered in any place of a line after a tab is positioned at the tab position.	Multilevel lists, embedded contents.

10.1.4.1.3. Advanced Text Style

The advanced text style is used for texts entered into the columns of the document title block or into the cells of other tables with fixed dimensions. The advanced text style sets relationship between the number of text lines: one, two or three and its parameters: character height, character spacing and line spacing.

The advanced text style is customized in the dialog (Fig.10.1.43), which appears after clicking the **Next...** button of the Text Style Creation and Editing dialog.

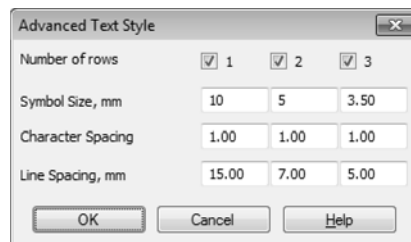


Fig. 10.1.43. Text Advanced Style Setup Dialog

The advanced text style is used only in tables, mainly for description of different cases of text input in the columns of the document title block, into a cell of the table with fixed dimensions, etc. When using a text style containing settings of an advanced text, the advanced style parameters are ignored outside tables.

For example, you can create text style *Normal* with 3.5 mm character height and adjust the advanced text style within text style *Normal* as follows: symbol size in a single line text— 10 mm; in a two-line text— 7 mm and in a three-line text— 5 mm. It means that the height of text characters in a table cell, to which *Normal* style was applied, will equal 10 mm if the text consists of a single line, 7mm if the text consists of two lines, 5mm if the text consists of three lines, and 3.5mm if the text contains four or more lines. For example, if *Normal* style is applied to the text of a text document, then regardless of the number of rows the symbol size will be equal to 3.5 mm.

Control elements of the Advanced Text Style Customizing dialog are represented in Table 10.1.19.

Tab. 10.1.19. Advanced Text Style Customizing dialog

Element	Description
Number of Rows	A group of options making it possible to specify the possible number of text lines.
Symbol Size	Fields for entry of the height of characters of the text consisting of a specified number of lines.
Compression symbols	The fields for entering character spacing in a text consisting of the specified number of lines.
Line Spacing	The fields for entering line spacing in a text consisting of a specified number of lines.

If a text consists of more lines than set when the advanced text style was customized, text options are ignored.

10.1.4.2. Practice in Creating and Editing Text Styles

This Section contains exercises for mastering the technique of creating custom text styles and editing system text styles.

Exercise 11. Creating a text style library. Independent work

Task. Create a text style library `user_texts.lts`.

1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Text Styles...**
2. Create a text style library with the name `user_texts` in the same way as you created a line style library in Exercise 1.

Without closing the Text Style Set and Text Style Library Operation dialog, proceed to doing the next exercise.

Exercise 12. Creating header styles and body text style

Task. Create text styles allowing formatting the documents in accordance with a layout model shown in Fig. 10.1.44. Non-italicized Arial Bold font must be used in styles.

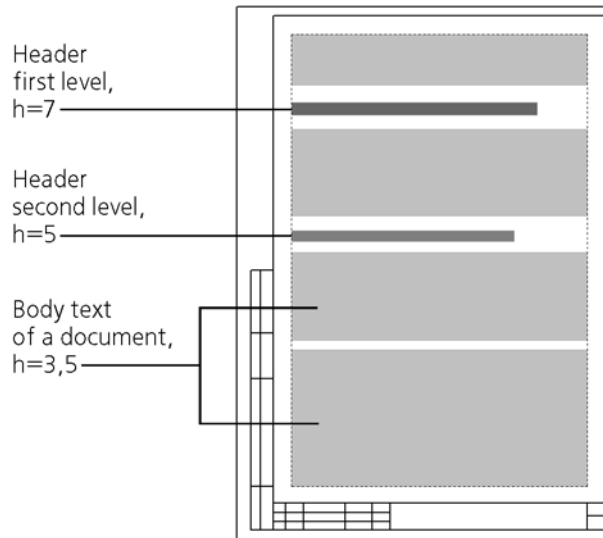


Fig. 10.1.44. Page layout



1. Click the **Create** button in text style sets and libraries dialog.



Make sure that the *user_texts.its* library window is active. Otherwise, a new style will be added to a library (a set or a document) opened in a neighboring window.

A Text Style Creation dialog will appear on the screen.

2. Enter value *100* into field **Number**.
3. Enter the name of a style to be created— **Body text** in the *Name* field.
4. Enter **5** in the *Line Spacing* field.
5. Enter **0** into the *First Line Indent* field as, according to the layout model, the first line of a paragraph does not have any indent.
6. Enter **0** into the fields of the *Indentation* group too, as the paragraphs of the body text begin and end directly at the entry border.
7. Enter **2** into the fields of the *Spacing* group.
8. Select the **Entire Width** option in the **Alignment** group, as the paragraphs of a body text are aligned at both sides (neither paragraph border is “ragged”).
9. Click the **Font...** button to set font options for a normal text.

A Font Options Customizing dialog will appear on the screen.

- 9.1. Select the **Arial** option from the **Font** list.
- 9.2. Enter or select **3.5** in the *Size* field.
- 9.3. Enter or select **1** in the *Spacing* field.
- 9.4. Click the **Color...** button and select black color for the text.
- 9.5. Do not activate the **Italics**, **Bold**, **Underlined** options.

- 9.6. Close the Font Options Customizing dialog by clicking the **OK** button.

A Text Style Creation dialog will remain on the screen.

The option **Advanced Text Style** allows setting of some text options, depending on the number of lines in it. This setting is used for creating styles for texts located in the table cells. In this case, advanced text style options need not be set.

10. Here the creation of the *Body text* style is completed. Close the Style Creation dialog by clicking the **OK** button.

A Style Set and Style Library Operation dialog will remain on the screen. The name of the style created by you — *Body text* — is displayed in the active preview window.



11. Click the **New** button.

A Text Style Creation dialog will appear on the screen.

12. Enter value *101* into field **Number**.

13. Enter the name of a style to be created, *Header 1*, in the **Name** field.

14. Enter **10** in the *Line Spacing* field.

15. Enter **0** in the field *First Line Indent*.

16. Enter **0** into the fields of the *Indentation* group too.

17. Enter the following spacing values: **Before Paragraph** — *12*, **After Paragraph** — *8*.



When setting a spacing, you must remember that a spacing between neighboring paragraphs is formed from the spacing after the preceding paragraph and the spacing before the following paragraph.

18. Select the **Left** option in the **Alignment** group. Selection of the **Entire Width** option is not recommended, as it usually results in disproportional increase of spaces between the header words, the paragraph of which consists of several lines (Fig. 10.1.45).



Fig. 10.1.45. Alignment of the header paragraph: a) to entire width b) to the left.

The alignment of body text paragraphs by width is also achieved by increasing the spaces between words. But due to the smaller size of characters and the presence of hyphenation, the lines of the body text contain more words than the header lines. Therefore, the difference between the spaces in the body text lines is inconsiderable.

19. Click the **Font...** button to set font options for the first level header.

A Font Options Customizing dialog will appear on the screen.

- 19.1. Enter or select **7** in the *Height* field.

- 19.2. Customize the rest of the options in the same way as you did for a body text (see items 9.1, 9.3–9.5).
- 19.3. Close the Font Settings dialog.

A Text Style Creation dialog will remain on the screen.

Do not activate the **Advanced Text Style** option, as in the case of customizing of a body text.
20. Here the creation of the *Header1* style is completed. Close the Style Creation dialog by clicking the **OK** button.

A Style Set and Style Library Operation dialog will remain on the screen. Two names of styles are displayed now in the active preview window — *Body Text* and *Header1*.
21. Independently create a style for the second level headers in accordance with a layout model.

Exercise 13. Check of text styles created

Task. Check whether the text styles created by you — **Header1**, **Header2** and **Body Text** — are customized correctly.

1. Create a new KOMPAS-3D text document.
2. Enter 6 paragraphs of a text of medium length (so as they, for example, would occupy half a page). A system text style is applied by default to all paragraphs.
3. Select the first paragraph.
4. Apply to this paragraph the *Body Text* style from the *user_texts.its* library created by you.
 - 4.1. Invoke the **Format— Style...** command.

A Current Text Style Selection dialog will appear on the screen.
 - 4.2. Click the **Library...** button in the **Show** group.
 - 4.3. In the dialog that appears on the screen, select the *user_texts.its* file and click the **Open** button.

The full name of the library selected will appear in the Current Text Style Selection dialog. The list of styles contained in it will be displayed on the **Library** page.
 - 4.4. Select the *Body Text* style in the list and click the **Select** button.

The Current Text Style Selection dialog will close. Make sure that the layout of the paragraph selected conforms to settings of the *Body Text* style: Arial font, capital letter size — 3.5mm, alignment by width.
5. Similarly, apply the *Header1* style to the second paragraph, and the *Header2* style to the fourth paragraph.
6. Apply the *Body Text* style to the remaining paragraphs.
 - 6.1. Select the required paragraph and invoke the **Format— Style...** command.
 - 6.2. Activate the **Library** page of the dialog which appears.

This contains a list of links to the library styles already used.
 - 6.3. Select the *Body Text* style in the list and click the **Select** button.

The Current Text Style Selection dialog will close, and the selected paragraph will be properly formatted. Thus, the activation of the library is not required for repeated selection of a style from this library.

7. Make sure that
 - ▼ Character size (vertical size of capital letters) and line spacing (distance between base lines of neighboring lines) in all paragraphs conform to the parameters specified in the process of customizing styles,
 - ▼ Alignment of paragraphs corresponds to the options selected when styles were customized,
8. Save the document as *text.kdw*.

Exercise 14. Editing of system text styles in the current document

As you know, system line styles, hatches, and texts are supplied with the KOMPAS-3D system. While editing of system styles of lines and hatches is limited (a user can change only the color and width of system lines, which, in turn, affects the view of the hatches used by these lines), system text styles allow for a very large range of editing options.

The second difference between system lines (hatches) styles and system text styles is as follows. The settings of system lines and hatches are saved in the system and applied to all open documents at the same time. The settings of system text styles are saved in the documents and, therefore, may differ in different documents.

Task. In the document *text.kdw*, edit the system text style for table headers. Use a non-italicized Arial Bold for the style. Character size — 3.5 mm. The text in a cell must be left-aligned.

1. Open the *text.kdw* document created in the previous exercise.
2. Create an arbitrary table in it with a header in the first line and fill the table cells. The texts in the table header cells have a *Table Header* system style with default settings.
3. Exit the table editing mode.
4. Call command **Service — Parameters.... — Current Text Document — Table heading**.
The right part of the dialog that appears on the screen contains control elements which allow to customize a *Table Header* text system style.
5. Enter **5** in the *Line Spacing* field.
6. Enter **1** into the fields of the *Offsets* group. Indents are distances from the text in a cell to the left and right borders of such a cell.
7. Enter the values in the **Spacing** group as follows: **Before Paragraph** — **2**, **After Paragraph** — **0**.
8. Select the **Left** option in the **Alignment** group.
9. Click the **Font...** button to customize font options for a table header.
A Font Customizing dialog will appear on the screen.
 - 9.1. Select the **Arial** option from the **Font** list. Then the **Different from Default Font** option is activated. Let this option remain activated. Deactivation of this option makes it possible, when customizing a style, to quickly select the font set for this document as the default font.



The default font for the current document is also selected in the Options Customizing dialog. To do this, you must select the **Default Font** option in the left part of the **Current Text document**.

In this case, the stated customizing is not required.

- 9.2. Enter or select **3.5** in the *Size* field.
 - 9.3. Enter or select **1** into the *Spacing* field.
 - 9.4. Activate the **Bold** option and deactivate the **Italics** and **Underlined** options.
 - 9.5. Clicking the **Color...** button, set a black color for the text.
 - 9.6. Close the Font Customizing dialog.
10. Customizing of a *Table Header* system text style is completed. Close the Options Customizing dialog by clicking the **OK** button.

You will see that the format of the first line of the table has changed: the appearance of the text and its location in cells correspond now not to the default settings of the *Table Header* style, but to the settings specified by you.

The style of the text in other cells will not change as it was not edited.

11. In the *text.kdw* document edit independently a system text style for table cells. Use non-italized Arial Bold font in the style. Character size — 3 mm. The text in a cell must be left-aligned.
12. Make sure that after changing a *Table Cell* system style, the appearance of the text in table cells has changed in accordance with the set parameters.
13. Save the document. In doing so, style settings made by you will be also saved. They will be used for table headers and cells in this document when such document is opened again, including at other workstations.

Exercise 15. Editing system text styles in new documents

Task. Edit the **Table Header** and **Table Cell** system text styles, intended for default use in new text documents. Styles must meet the same requirements as the styles with the same name in the *text.kdw* document.

1. Call command **Service — Parameters... — New documents**.
2. Open the **Text Document** section in the right part of the current dialog.
3. Select the **Table Header** option in this section.
The right part of the dialog will display control elements for customizing the *Table Header* system text style, which will be used by default in new text documents.
4. Do Items 5–9 from Exercise 14.
5. Select the **Table Cell** option in the **Text Document** section, and similarly customize the system style of the same name.
6. Close the Options Customizing dialog by clicking the **OK** button.

Exercise 16. Check on customization of system text styles.

Task. Check system text styles **Table header** and **Table Cell** in a new text document for proper default settings.

1. Create a new text document.
2. Call command **Service — Parameters.... Current text document**.
3. Sequentially selecting the **Table Header** and **Table Cell** options in the left part of the dialog, make sure that the new document has acquired all required settings of the text style with the same name.

You can also create a table in the new document, and after filling its cells check whether a text has acquired by default the required options.

Thus, if the text styles used by you differ from default styles, you must perform the following operations.

- ▼ Customize the system text styles for new documents, so that they have the required options.
- ▼ Create the missing text styles (for headers of sections and subsections, captions for illustrations, etc.) and save them in a custom text style library.

10.2. Document Layouts

10.2.1. Title block and document layouts

Layout is assigned to pages of each document — drawing, text document, BOM.

By default, a drawing's pages have the layout corresponding to system configuration. For example, the first page of the KOMPAS-3D mechanical engineering drawing has the following layout: *Design Drawing. First Sheet. GOST 2.104-2006*, while its subsequent pages have the following template: *Design Drawing. Final pages. GOST 2.104-2006*. While working with a drawing, you can assign another template to any page of this document. See Section 3.5.2.1 on p. 1151 for more details about changing the drawing's layout.

When working with a text document, its default layout can be changed. Layout is assigned to the first page. For all other pages, it is assigned automatically to all even or add pages. See Section 4.1.5.2 on p. 1343 for more details about changing the text document's layout.

Layout is an integral part of the BOM's style affecting its completion. For more information about changing BOM style, refer to Section 6.3.4 on p. 1561.

10.2.1.1. General Information on Layouts and Title Blocks

Layout is a part of a drawing or a text document of KOMPAS-3D system including a Title Block and borders (external and internal). The Layout of text document also include info about the Text Input Field dimensions, and the drawing Layout include info about Revision Table availability and the rules of its location on the sheet.

Title Block includes one or several tables, the cells of which have special settings. These settings make available some service functions (automatic data input, data transfer from one cell to others, etc.).

The using of Layouts significantly facilitates creation of typical documents: it means that you are not required to draw out new borders and tables of the Title Block as they are already included into the Layout assigned to this document.

KOMPAS-3D includes standard Layouts for design documents. These Layouts are stored in the System Layout Library: file *Graphic.lyt* that is stored in subfolder `\Sys` of the main system folder.



The package also includes file *Vector.lyt*. This file contains the Layouts of documents to be printed using vector devices.

To provide operation of some libraries, e.g., SPDS Designations library, some Special Layouts, different from the system Layouts, are required. These Layouts are included into the libraries packages to be supplied. When installing these libraries the corresponding Layout Libraries will be placed in subfolder `\Sys`.

User can edit the existing Layouts and create new ones. Creation of custom Layout Libraries is also possible. These libraries can include both new and existing Layouts, i.e., the Layouts can be copied between the libraries.

As long as the Title Blocks are the components of the document Layouts, they are stored in the same libraries *.lyt as the Layouts. The Title Blocks are created and edited as separate objects. The Layout contained in any library *.lyt can include only the Title Block stored in the same library.

Title Blocks and Layouts of the drawings and text documents are handled using the dialogs called by the commands **Service— Libraries of Styles, Layout Types and Title Blocks— Title Blocks...** / **Drawing and BOM Layout...** command / **Template of Text Documents....** The control elements of these dialogs are practically analogous to that of the dialogs for use of object style library sets (see 10.1.2 on p. 1910). The difference is that instead of the group of three buttons— **Show Set, Show Library, Show Document**— it has only one button— **Show Library** because the Title Blocks and Layouts, contrary to styles, can be stored only in libraries.



When copying a Layout from one library to another, the Title Block included into this Layout is also copied.

You can assure yourself of this performing Exercise 18 from Section 10.2.2.

The dialog of the Drawing Layout Library use is shown on Figure10.2.1.

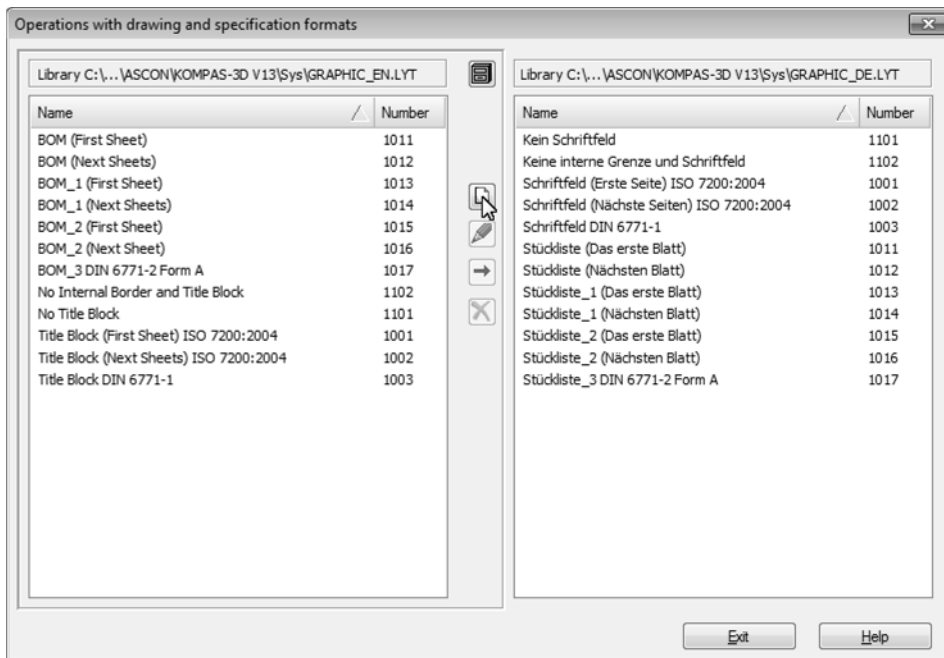


Fig. 10.2.1. Dialog of drawing layout library use

10.2.1.2. Storing data in documents about used layout

The design assigned to the document (together with the included title block) is embedded into the document. Therefore, the format frame and the title block are displayed in the document, even if *.lyt library containing the document's design is not present.

In addition, a link to an external library file *.lyt containing the selected layout is stored in the document. The link includes path to the library and its number of layout. Therefore, when design is changed in the library, the document's design can be updated.

You can enable automatic layout update when customizing KOMPAS-3D (see Section 10.2.1.2.1).



All links available in the current document, including the links to the Layout Libraries, can be viewed on the page **External Links** of the document summary dialog. Use the **File — Document summary...** command to activate this dialog.



Layout stored in the library can be edited (see Sections 10.2.1.5 on p. 1965 and 10.2.1.7 on p. 1978), layout embedded to a document— can not.



Starting with the 13th version of KOMPAS-3D layout is embedded to a document. If the document had been created in an earlier version, absence of *.lyt library containing its design results in its opening without the frame and the title block.

10.2.1.2.1. Layout Update

Design update — replacement of the design embedded into the document with a design whose number and path to the library are saved in this document.

The design update does not affect the texts entered when filling in the title block; these texts will remain unchanged in the document.

Layout update customization is made in the dialog activated by the **Service — Settings... - System - General - Document Layout Update**.

- ▼ You should select the **Renew** option for automatic update in this dialog. Additionally the **Request Confirmation** option can be enabled.

In this case, if differences are found between the design saved in the document and the respective design in *.lyt¹ library, the system will do the following:

- ▼ If confirmation request is disabled, layout is updated automatically. The pop-up message about applied update appears on the screen (Fig.10.2.2, a).

1. Designs are always compared when opening a document. Once the library has been changed, designs are compared immediately, if the library is edited on the same working place where the document is opened. If the document is opened on another working place, designs are compared in accordance with access rights settings (see Section 9.1.4.2 on p. 1694).

- ▼ If the confirmation request is on, the message illustrated in Fig. 10.2.2, b will appear on the screen. Clicking **Yes** means layout update, **No** — no update.

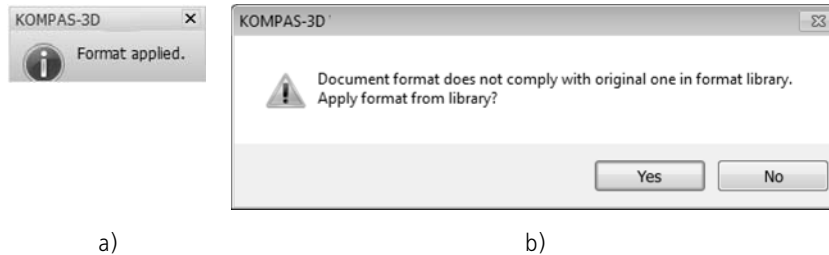


Fig. 10.2.2. Messages: a) on finishing layout update,
b) on the need to update design,

- ▼ To cancel an automatic update, the **Do Not Update** option must be chosen in the update settings dialog. Additionally, the **Report about layout changing in the library** option can be enabled.

In this case, work with layout documents is performed in the following way:

- ▼ If messages informing about changes of designs in the library are disabled, no messages about document designs will be appearing.
- ▼ If messages informing about changes of designs in the library are enabled, pop-up messages will be appearing to inform about differences between the design saved in the document and the respective design in *.lyt library. Message also contains the **Apply Layout from Library** command (Fig.10.2.3). You can update layout by calling this command.

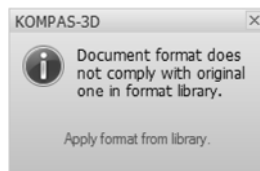


Fig. 10.2.3. Prompt to update design



The design of the current document will updated in accordance with the setting applicable on the moment the document was being opened. Therefore, to apply the layout update changes (for example, enabling of automatic update) it is necessary to close documents and reopen them.

If layout update is not possible, the respective message will be displayed on the screen. Message examples are shown on Fig.10.2.4. If design update is enabled, the message appears every time the update takes place. If design update is disabled, the message appears only if messages informing about design change are enabled.

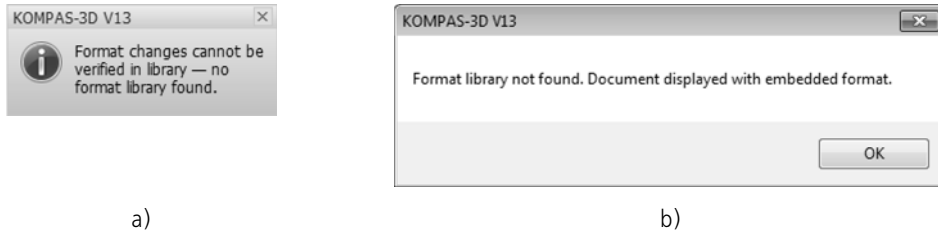


Fig. 10.2.4. Messages: a) no style in the library, b) no library

A forced design update is possible at any moment when working with a document, see Section 10.2.1.2.2 on p. 1963.



If a document's design is different from the respective design in **.lyt* library, this document design name is preceded by an asterisk (*) in all dialogs where it may be displayed (for example, see Fig. 3.5.2 on p. 1151 in the Document Manager).



You cannot cancel the document design update by the respective command. Therefore, to restore earlier design, the document should be closed without saving and opened again.

10.2.1.2.2. Layout re-reading

Layout Re-reading — forced document layout update.

Layout re-reading is available:

- ▼ **For all document pages.** You do this using **Service — Reread Layout** command. This command is available if there are differences between layout stored in a document and respective layout in the **.lyt* library. If there are no differences and also if there is no library or no layout in it (i.e. it is impossible to compare layouts), the command is unavailable.
- ▼ **Arbitrary drawing sheet.** To do this, call from the title block context menu of desired sheet **Change Layout** command and click in the appeared dialog **Re-read** button. In addition, re-reading of specific pages layouts is possible in the **Document Manager** dialog (see Section 3.5.2.1 on p. 1151): select a row corresponding to a desired page in the pages list, click the mouse in the layout cell and click the **Re-read** button in the appeared dialog.
- ▼ **first page, all pages with even numbers, all pages with odd numbers, additional page in the beginning or additional page in the end of a text document.** To do this, call the **Service — Parameters... — Current Text Document — Sheet Parameters**.
 - ▼ To re-read the design of the first, even-numbered or odd-numbered pages, choose the **Design** item in the left part of the appearing dialog; then click the **Re-read** button in the right part for the required group.
 - ▼ To re-read design of an additional page, choose the **Additional pages** item in the left part of the appearing dialog; then choose design in the right part and click the **Change**; click **Re-read** in the next dialog.

Close the setup dialog by clicking the **OK** button.

10.2.1.3. Number of Title Block and Layout in the Library

Each Title Block and each Layout has its number, which is assigned to them when created.

The number is a unique attribute of the Title Block (Layout).

One and the same library cannot contain two Title Blocks with the same numbers. The reason is that the Title Block is included into a Layout just via its number.

One and the same library cannot contain two Layouts with the same numbers. The reason is that when assigning a Layout to a document, a link to this Layout will be included into the document. The link includes the following data: library full name and the number of a Layout from it.

Besides, the number of a Layout assigned to a document is important for compatibility with the previous versions, as well as for KOMPAS-3D application development using language tool kit. Numbers from 1 to 100 are reserved for standard Layouts, so when creating new Layouts it is recommended to use numbers starting from 101.



The numbers of Title Blocks and Layouts contained in the library can be the same or different — it does not matter. For example, the numbers of Title Blocks in library *Graphic.lyt* coincide with the numbers of Layouts, which use these Title Blocks, but this is done only for usability.

At an attempt to put into a library (by creation or copying) the Title Block, the number of which coincides with the number of one of the Title Blocks in this library, a warning dialog will appear on the screen. This dialog is analogous to the dialog warning about the style existence in a library (see Fig.10.1.6 on p. 1916). This dialog also includes the titles of library objects (Title Blocks in this particular case) having the same numbers, as well as control buttons (see Table). 10.1.4 on p. 1916).

When inserting a new or copied Layout into a library, the presence of the copied Title Block in this library is checked first and then— the presence of the copied Layout.



If an object (Title Block or Layout) with the same number and the title as the object to be added exists in the library, a request to overwrite the object, i.e. to replace the old object with a new one, will appear on the screen. You can confirm or reject the overwrite.

If a probability of Layout exchange among several libraries is high enough, make sure that all Layouts (as well as Title Blocks) in these libraries have different numbers. It will help to prevent the conflict situations described above.

10.2.1.4. Recommendations on Document Layout Creation

The process of the document Layout creation includes two main steps:

1. title block creation,
2. Layout creation using this Title Block.

You can create your own library **.lyt* to store a new Title Block and Layout. The process of Layout Library creation is analogous to the Style Library creation (see Section10.1.1.2.1 on p. 1911).

If a newly created Title Block (Layout) is a modification of an existing object, then this existing object can be used as a prototype (see Section 10.2.1.6.2 on p. 1976).

Prior to proceed to the Layout creation, it should be thought over carefully. Particularly, it refers to the Title Block tables, as their cells should be set up correctly (see Sections 10.2.1.5.1, 10.2.1.5.2).

It is recommended to test the created Layout carefully and then immediately correct the detected errors. It is due to the fact that the introduction of alterations into Layouts that have been already assigned to documents usually takes a lot of time.

Considering that the document Layout is a relatively complex object, special skills are required to handle it. That's why it is recommended to get acquainted with Section 10.2.1 and its subsections, and create several trial Layouts performing the Exercises in Section 10.2.2, prior to creation of the Layouts to be used for documentation issue in your Company.

10.2.1.5. Title Block Creation

A Title Block being a part of the document Layout is a special compound object of KOMPAS-3D. The Title Block can include one or several tables. These tables have only one dissimilarity from usual tables. It consists in the fact that during creating of the Title Block table each cell should be set up in a special way (see Section 10.2.1.5.2 on p. 1968). The fill-in process for a document Title Block table is the same as the fill-in process of a usual table.

The general procedure to create a new Title Block is as follows.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Title Blocks...**
2. Open an existing library or create a new library, where a new Title Block will be included.
3. Click button **Create** to proceed to creation of a new Title Block in the current library.

The Title Block creation dialog will appear on the screen (Fig. 10.2.5).

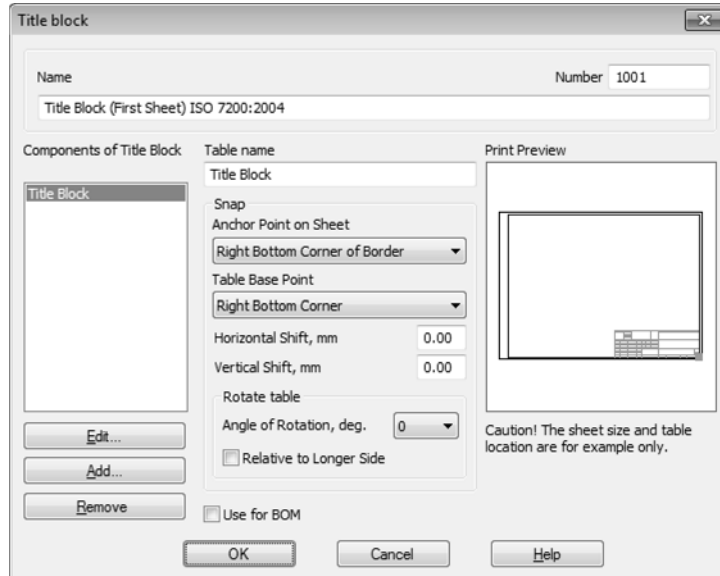


Fig. 10.2.5. Title block creation and editing dialog

4. Enter the name and the number of a new Title Block. You can use any name for the Title Block. Usually, it contains brief information about the Title Block (e.g., standard and application area). The rules of Title Block numeration are specified in Section 10.2.1.3 on p. 1964.
5. Click button **Add** to proceed to creation of the first table of the Title Block.
6. Create and set up the first table of the Title Block (see Sections 10.2.1.5.1, 10.2.1.5.2).
7. Save the created table and close its window.
The system will return to the Title Block creation dialog.
The name of a newly created table (it is displayed in the Title Block creation dialog field having the same name)— «Unnamed».
8. Enter the name corresponding to the table purpose into field **Table Name**.
9. Set up location of the created table on the page in the Title Block creation dialog (see Section 10.2.1.5.3 on p. 1975).
10. If the created table is used to input BOM items, then enable option **Use for BOM**.
11. Add the required number of tables into the Title Block and set up their location repeating items 5–7 as necessary.
12. Close the Title Block creation dialog.
The system will return to the Title Block use dialog. The name of the created Title Block will appear in the list of Title Blocks of the current library.
13. Close the Title Block use dialog.



It is expected that the User proceeding to the Title Block table development possesses good skills to use tables in the documents of KOMPAS-3D system. That is why only specific options and options having a particular impact on the Title Block tables, but not all options, are described in Section 10.2.1.5.1.

10.2.1.5.1. General Table Setting Techniques

The preliminary setting up of a new table of the Title Block, as well as usual tables, is carried out using the new table creation dialog (Fig. 10.2.6). The dialog appears on the screen after clicking button **Add...** in the Title Block creation dialog (Fig. 10.2.5).

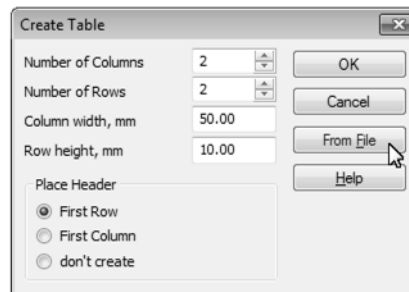


Fig. 10.2.6. Dialog for creating a new table

If the table for the Title Block was prepared in advance (see Section 10.2.1.6.1 on p. 1976), click button **From File** and open the required table or fragment file.

If the table was not prepared in advance, set its parameters and click **OK**.

The system will go into the table editing mode: a new window with the table of set parameters will appear on the screen. Main Menu, Compact Panel and Property Bar will display in this mode the commands to work with tables.

When creating tables of title block, all methods of working with tables in KOMPAS-3D documents will be available (see Section 4.2.1.1 on p. 1349). These options allow to adjust **General Settings** of the Title Block table. In addition, each cell of this table should necessarily have a **Special Setting**— an advanced format setting (see Section 10.2.1.5.2). This option is not available when creating usual tables.

When adjusting general settings of the Title Block table, the following information should be taken into consideration.

- ▼ An automatic ban to edit the text will be set for each table cell containing any text. In other words, the presence of even one symbol in the cell means that it will be impossible to edit the text in this cell when filling in the Title Block in the document. The User will be able neither to delete the existing text, nor to supplement it, nor to change its parameters (font, color, size, etc.).
- ▼ The appearance of the text, which will be entered into an empty cell when filling in the document Title Block, is determined by the formatting applied to this cell during creation of the Title Block table. To set the cell format place cursor into this cell and click command **Format— Cell**

Format... In the appeared dialog, you can set various parameters: text style, text offsets from cell borders, etc.

Note that the text style can have advanced style settings— info about the required line spacing, symbol size and spacing in the text depending on the number of lines in this text (see Section 10.1.4.1.3 on p. 1951).

If the custom style should be applied to the text in a cell, it is recommended to save the style directly in the Title Block to be created— this will exclude style loss when moving the library to another work station. Text style creation in the Title Block is analogous to text style creation in the document (see Section 10.1.1.2.2 on p. 1912).

- ▼ Once the cell dimensions and text offsets are determined, it is recommended to fix the table dimensions. To do this, call command **Table— Table Fixing** and enable all options in the appeared dialog.
- ▼ If the Title Block table to be created is located on the sheet in such a way that some of its borders coincide with the document internal border (or with the borders of other tables), it is recommended to select the following line style for these borders: *Hidden*.

It is due to the fact that some vector-type output devices process an even number of lines imposed on each other as 'no line'. As a result, some lines of the Layout will be absent in the printed document.



All in all, the cells of the created Title Block table should either **have the text**, or **have certain format and advanced format settings**, i.e., the settings determining the properties of the text to be entered into the cell when filling in the Title Block.

10.2.1.5.2. Cell Advanced Format Setting

Advanced Cell Format is a package of the Title Block cell properties, determining which service functions will be available when filling in the corresponding column of the Title Block in the document (automatic input of some data types, data transfer from one cell to other cells, calling of user menu, etc.). In addition, the cells properties specified during advanced format setting up determine the possibility and order of data transfer from one Title Block to another one when changing the document Layout.



The cell advanced format is set up in the dialog box having the same name (Fig. 10.2.7). It is called by clicking button **Next...** in the cell format setup dialog, or by clicking button **Advanced Cell Format** on panel **Tables and Borders** or on page **Tables** of Property Bar.

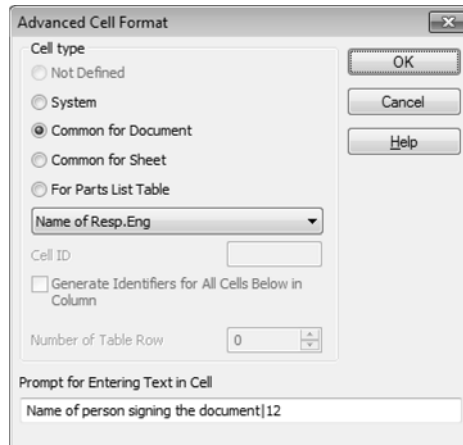


Fig. 10.2.7. Advanced Cell Format Settings dialog

When creating the Title Block table, the advanced format should be set up for each cell, which will be filled in by User during the work with the document. To do this, place cursor successively into empty cells of the table, call the cell advanced format dialog and set necessary properties of the current cell (where the cursor is placed) in this dialog.

Each empty cell of the Title Block is characterized by the following properties:

- ▼ **type**,
- ▼ **ID**,
- ▼ **Identifier value**.

If the advanced format setting of the current cell has not been defined yet, its type is considered to be unspecified, and, as a consequence, at the first calling of the dialog the option **Not Defined** will be enabled in it. The cell type can not be left unspecified as otherwise the data input into this cell when filling in the Title Block will be impossible.

Available variants **System**, **Common for Document**, **Common for Sheet** and **For BOM Table** of group **Cell Type** allow to select the data type of the current cell.

Cell type descriptions and identifier assigning rules are given in Table10.2.1.

Tab. 10.2.1. Cell types of the Title Block table

Type	Cell Description	Example
System	A cell of this type is filled in automatically (without User participation) in the document Title Block. Select the required identifier from the list to define which data is to be entered into a cell of system type.	Columns <i>Sheet number</i> and <i>Number of sheets</i> of main table in text documents and BOM's.

Tab. 10.2.1. Cell types of the Title Block table

Type	Cell Description	Example
Common for Document	<p>The data entered into a cell of this type, when filling in the Title Block, can be automatically transferred into other cells of the Title Block tables on the same sheet and on other sheets of a multisheet document.</p> <p>To transfer data it is required that cell types and identifiers coincide.</p> <p>When changing the document Layout, the data from this cell is transferred into the cells of a “new” Title Block, which have the same type and identifier.</p> <p>Accordingly, type Common for Document should be set up and the same identifier should be selected for the cells of all Title Blocks, between which the data should be transferred automatically.</p>	<p>Column <i>Designation</i> of the main table, where the document designation is recorded. The text can be entered on any sheet of the document and this text will be automatically entered into <i>Column 26</i> of the same sheet and analogous columns of all other sheets in case of a multisheet document.</p>
Common for Sheet	<p>The data entered into a cell of this type, when filling in the Title Block, can be automatically transferred into a cell of another table of the Title Block on the same sheet.</p> <p>To transfer data it is required that cell types and identifiers coincide.</p> <p>When changing the document Layout, the data from this cell is transferred into the cells of a “new” Title Block, which have the same type and identifier.</p> <p>The content of this cell is not transferred into all sheets of a multisheet document.</p>	<p>Column <i>Revision</i> of the main table, where the document revision number is recorded. When changing one type of the Title Block of this document for another type, the column content will be automatically transferred into an analogous column with the same identifier.</p>

Tab. 10.2.1. Cell types of the Title Block table

Type	Cell Description	Example
For BOM Table	<p>This type is intended for BOM table cells (the Layout containing the Title Block with such table is included then into the BOM style). When setting this cell type it is required to select the following identifiers from the list: Ref. No., Name, Designation, Number, etc.) and specify the number of BOM table row, where the cell to be set up is located. All cells of the same column should have the same identifiers and differ only by the row number. Use option Generate Identifiers for All Cells below in Column to speed up the setup process for column cells of the same type (assigning identifiers and row numbers to them).</p>	

It is necessary to set value for the identifier in field **Cell ID** for cell type **Common for Sheet**. Identifiers of cells of all other types should be selected from the drop-down list (Tables 10.2.2 and 10.2.3). The values of these identifiers are predefined, therefore, field **Cell ID** will be inaccessible for data input.

Identifier **Custom** is one of the predefined identifiers. The peculiarities of its use are described in Section *Peculiarities of Using Cells of Types Common for Sheet and Common for Document* on p. 1973.

Tab. 10.2.2. Identifiers of System Type Cells

ID	Identifier values
Sheet number	7
Number of Pages	8
Format	Defined automatically by system
File Name (with Path)	43
File Name (Proper)	44
Designation line and Hyphen	45
Custom	

Tab. 10.2.3. Identifiers of 'Common for Document' Type Cells

ID	Identifier values
Product Name	1
Document Designation	2
Material designation	3
Document letter (Column 1)	40
Document letter (Column 2)	41
Document letter (Column 3)	42
Mass of Product	5
Scale	6
Manufacturer's code	9
Name of Resp. Eng.	110
Dev. finish date	130
Name of Check. Person	111
Checking date	131
Name of Mfg. Eng.	112
Process inspection date	132
Work type	10
Name of Draw. Person	113
Drawing date	133
Name of Qual. Eng.	114
Qual. insp. date	134
Name of Appr. Person	115
Approval date	135
Designation of Variant [Main Variant]	2110
Designation of Variant [Var.1]	2111
Designation of Variant [Var.2]	2112
Designation of Variant [Var.3]	2113
Designation of Variant [Var.4]	2114

Tab. 10.2.3. Identifiers of 'Common for Document' Type Cells

ID	Identifier values
Designation of Variant [Var.5]	2115
Designation of Variant [Var.6]	2116
Designation of Variant [Var.7]	2117
Designation of Variant [Var.8]	2118
Designation of Variant [Var.9]	2119
Document Number	Not used
Document Name	51
Document Code	52
OKP Code	53
Custom	



Identifier values are displayed in the Message line in the Title Block table editing mode.

The text can be also entered in the cell advanced format setting dialog, this text will be displayed in KOMPAS-3D message line when filling in this cell of the Title Block (when cursor will be inside the cell). This text should be entered into field **Prompt for Entering Text in Cell**.



The prompt line should explain to the maximum extent the purpose of the current cell, as it will serve as the only system tip when entering the text into this cell.

If it is required that the User Menu will be available when entering the text into this cell, then symbol “|” (vertical line) followed by the Section number of the User Menu file, the rows of which should be displayed in the Title Block column to be filled in when calling out this menu, should be entered directly after the prompt text.

Text input into the Title Block columns by selecting the required rows from the User Menu speeds up the fill-in process and excludes misprints, which can occur while typing. The purpose of User Menu file, its structure and syntax are described in detail in Section 9.5.5.1 on p. 1890

Peculiarities of Using Cells of Types Common for Sheet and Common for Document

The using of types **Common for Document** and **Common for Sheet** on one and the same sheet gives the same results - the data entered into the cells of this type can be transferred (provided that identifiers coincide) into the cells of other tables on the same sheet. However,

type **Common for Document** is more universal as it provides also the data copying between the sheets in a multisheet document. Because any of the Title Blocks can be used for multisheet documents, it is recommended to adhere to the following rules.

- ▼ The cells containing the data, which are unique for this sheet, should have type **Common for Sheet**. If, among these cells, there are some cells, in which the data should be the same, then assign to them the same identifiers; this will ensure the automatic fill-in of other cells on this sheet upon filling in of any such cell.
- ▼ The cells containing the data, which are unique for this document, e.g., its designation, part name, manufacturer's identification), should have type **Common for Document**. If, among these cells, there are some cells, in which the data should be the same, then assign to them the same identifiers; this will ensure the automatic fill-in of other cells on sheet(s) upon filling in of any such cell.
- ▼ The cells, which are to be filled in by hand after the document is printed out, e.g., cells for signatures), can have either type **Common for Sheet**, or **Common for Document**. The only thing to check in this case is that no data is transferred to these cells from other cells. For a cell of type **Common for Sheet** it is achieved by assigning of a unique identifier value, and for a cell of type **Common for Document**— by assigning of identifier **Custom** with a unique value.

Use of Custom Identifiers

So, when setting up the cell advanced format it is required to define the cell type and its identifier. If there is no required identifier among the predefined, i.e., included into the list, ones, select variant **Custom** from the list.

In this case it is required to set value of the custom identifier to specify the cells, between which the data should be transferred. This value (integer number) is entered into field **Cell ID**, which becomes available upon selection of the custom identifier.



The values of custom identifiers assigned to the cells of different purpose should be necessarily different!

E.g., it is required to transfer data between the cells having custom identifiers and located in the Title Blocks of sheets in a multisheet document. For this purpose, it is necessary to assign type **Common for Document** to all these cells and set the same identifier values.

At the change of the document Layout the data transfer is possible between the cells having custom identifiers. A condition for such data transfer is a coincidence of types and identifier values of the cells of a previous Title Block and a new one.



Care should be exercised to see that the custom identifiers assigned to analogous cells have the same values, and it is required only when creating the Layouts, which can be interchanged with a high probability.

To avoid an accidental coincidence of an identifier assigned by User with an identifier already existing in the system (it is clear that it will lead to “mixture” of data in the cells) it is recommended to use identifiers specified in Table10.2.4.

Tab. 10.2.4. Recommended Identifiers (Values of Custom Identifiers)

Range	Cell Type
1000–2000, 2500–16383	Common for Sheet, Common for Document (Custom identifier)
100–150	For BOM

Identifier Control

When creating the Title Block table it is possible to check its cells for available identifiers and for their coincidence.

Call command **Service— Identifier Check** to carry out such check.

The identifier control dialog will appear on the screen. The current Title Block table is displayed in this dialog. Those cells of the table, the types and identifiers of which coincide, are highlighted with blue color, and those cells, the type (and identifier correspondingly) of which is not defined— highlighted with pink.



The presence of cells without identifiers or with the same identifiers is a gross mistake in the Title Block table creation. Therefore, if such cells were found, it is required to set their advanced format repeatedly.

10.2.1.5.3. Setting of Table Position on Sheet

To set the Title Block table position on sheet, select it in list **Components of Title Block** of the Title Block creation dialog. This Title Block will become the current one, i.e., its name will be displayed in field **Table Name**, and its image will be highlighted in window **Print Review**. The elements located in group **Snap** control the table position on sheet. These elements are described in Table 10.2.5.

Tab. 10.2.5. Elements of **Snap** Group

Element	Description
Base Point on Sheet	The list allowing to define, which characteristic point of the document internal border will be used as a base point for the current table snapping. This point is displayed in window Print Preview in the form of a small square.
Base Point of Table	The list allowing to define, which characteristic point of the table will be used as a base point for snapping.
Offset Horizontally, Offset Vertically	Fields to input the table base point offsets relating to the base point on sheet.

Tab. 10.2.5. Elements of **Snap** Group

Element	Description
Table Rotate	<p>A group of elements determining the value and direction of the table rotation around its base point.</p> <p>The rotation value is set up by selecting the required row from list Rotation angle. The list includes the angle values divisible by 90°.</p> <p>In case the table location depends on a sheet format and orientation, e.g., Table <i>Column 26</i> of the Title Block, it is necessary to enable option Relative to Longer Side.</p>

10.2.1.6. Advanced Methods of Title Block Creation

Some steps of the Title Block creation can be sufficiently expedited using such methods as table preliminary preparation and prototype use.

10.2.1.6.1. Preliminary Preparation of Tables

Prior to creation of a unique Title Block (with no existing prototype) it is possible to form preliminary all tables, which will constitute the Title Block, and save them in separate files on the hard disk.



The preparation of table files is not obligatory. The table creation is possible during the Title Block formation.

Carry out the following steps to create a table file.

1. Create a graphic or text document in KOMPAS-3D.
2. Create a table in this document, arrange it and set the required format for its cells.
3. When in the table editing mode, call command **File— Save Table to File...**
4. Select a folder and enter a file name to save the table in the appeared dialog box. The default extension of table files in KOMPAS-3D — *tbl*.



If the table of created Title Block practically has no areas with regular structure, it is more reasonable to pre-draw it in a fragment of KOMPAS-3D(*.frw).

10.2.1.6.2. Use of Title Block Prototypes

If some Title Blocks already exist in the current library, you can use any of them as a prototype for a newly created Title Block. To do this, prior to clicking button **Create** in the Title Block use dialog, select the required Title Block in the list (Fig.10.2.8).

Reply “Yes” on a system question about using the selected Title Block as a prototype (Fig.10.2.9).

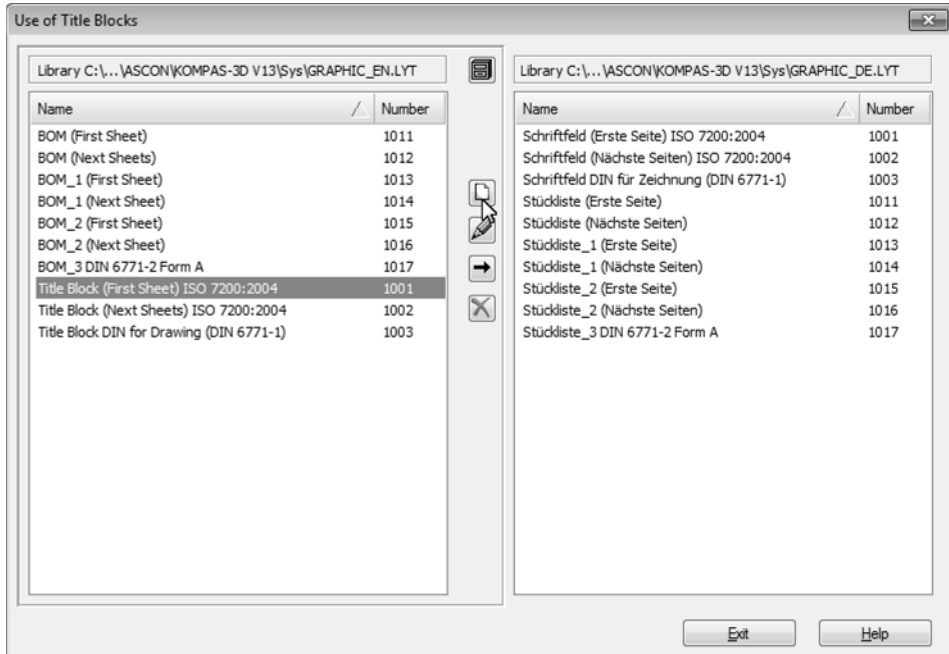


Fig. 10.2.8. Selection of Prototype Title Block

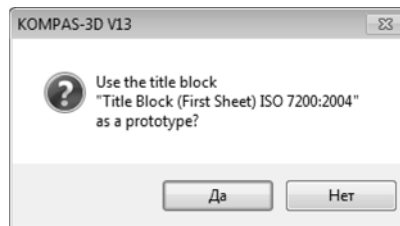


Fig. 10.2.9. Prototype Use Confirmation

In this case the Title Block creation dialog will display the list of the prototype Title Block tables (Fig.10.2.10) and you will be able to develop a new Title Block editing these tables and their location. Table deletion and addition of new ones is also possible.

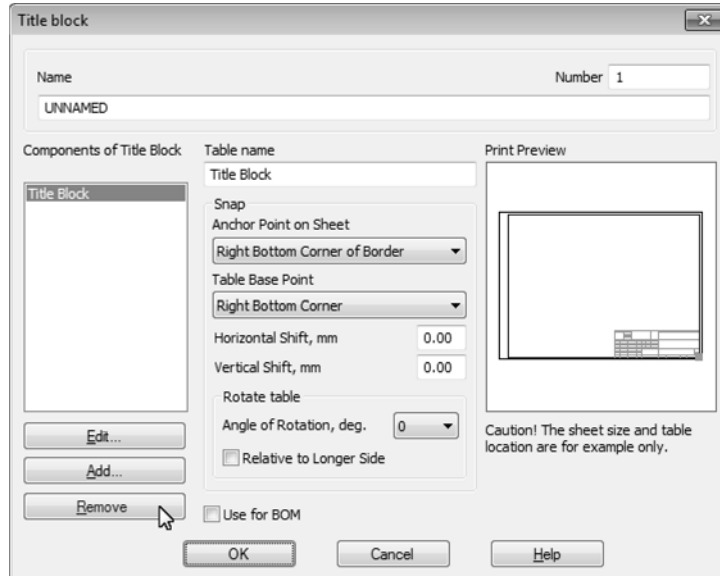


Fig. 10.2.10. Prototype Title Block Editing

To edit the table selected from the list click button **Edit...**, and to delete it — button **Delete**. The Title Block created using a prototype is not different from the Title Blocks created without it.

In a similar way Layouts can be created based on a prototype.

10.2.1.7. Layout Creation

After setting up the Title Block, which will be included into a new document Layout, it is possible to proceed to creation of the Layout itself.

The general procedure to create a new Layout is as follows.

1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Drawing and BOM Layout... Layout of Text Documents...** depending on the type of the document, for which this Layout is created.
2. In the appeared Layout Use dialog box open library **.lyt*, which contains the Title Block to be used in the created Layout.



The Title Blocks stored in a current library **.lyt* are **not displayed** in the Layout Use dialog. To open the required library, you should know its file name. E.g., it can be memorized or recorded when creating the Layout.



3. Click the button **Create** to proceed to creation of a new Layout in the current library.
4. Set up the properties of a new Layout in the appeared dialog (see Section 10.2.1.7.1). Close the Layout Setup dialog by clicking button **OK**. The system will return to the library use dialog. The name of a newly created Layout will appear in the list of Layouts of the current library.

5. Close the Layout Library use dialog.

10.2.1.7.1. Layout Setting

Drawing and text document Layout setting up is carried out in the dialogs with the same names. The dialogs to set up drawing Layouts (Fig.10.2.11) and text document Layouts (Fig.10.2.12) differ from each other partially. The elements common for both dialogs are presented in Table10.2.6.

Tab. 10.2.6. Common Control Elements of Document Layout Setup dialogs

Element	Description
Name	A field for layout name input. Usually, the name includes brief information about the Layout purpose.
Number	A field for layout number input. The Layout numbering rules are given in Section10.2.1.3 on p. 1964.
External Border	An option controlling the presence of an external border limiting the format in the Layout. Select the required row from list Line to define the line style for external border drawing. The current style line appearance is displayed in a preview window to the right from the list.
Internal Border	An option controlling the presence of the internal border in the Layout. Select the required row from list Line to define the line style for external border drawing. The current style line appearance is displayed in a preview window to the right from the list. Enter the required values into the fields of group Clearance to External Border to set the distance between the corresponding sides of the external and internal borders.
Title Block	An option controlling the presence of the Title Block tables in the Layout. The name and number of a Title Block used in the Layout to be set up is displayed in the field located under this option. Click button Select located to the right from the field to change the Title Block. A dialog containing the list of the Title Blocks included into a current library *.lyt will appear on the screen.



It is more convenient to select the Title Block if its name coincide with the name of the Layout to be set up, e.g., like in library *Graphic.lyt*.

The drawing Layout Setup dialog additionally includes the group of elements **Revision Table**. These elements are presented in Table 10.2.7.

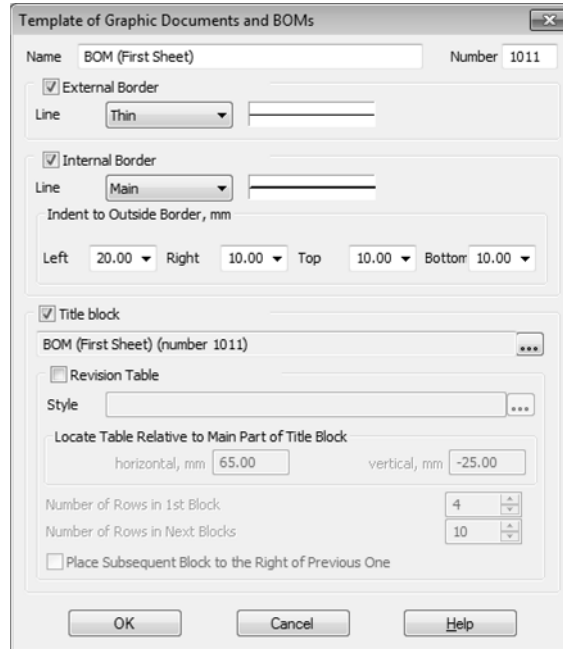


Fig. 10.2.11. Drawing Layout Setup dialog

Tab. 10.2.7. Group of Elements **Revision Table**

Element	Description
Revision Table	An option controlling the presence of the Revision Table in the Layout as a separate table not being a part of the Title Block. Such Revision Table is developed using BOM design module, i.e. it is actually a specifically set up BOM table.
Style	A field, where the Revision Table style, used in the Layout to be set up, is displayed. Click button Select located to the right from the field to change the Revision Table style. A dialog containing the list of the BOM styles included into a current library *.lyt will appear on the screen. Here it is necessary to select a style corresponding to the Revision Table, which will be used in the Layout to be set up*. Two styles developed for the Revision Table are supplied with KOMPAS-3D system: <i>Revision Table. GOST 2.104-2006</i> and <i>Revision table with zones. GOST 2.104-2006</i> . They are stored in library <i>Graphic.lyt</i> .

Tab. 10.2.7. Group of Elements **Revision Table**

Element	Description
Table Offset relative to Main Table of Title Block	Fields to input an offset of the Revision Table lower-right angle in relation to the Title Block main table upper-left angle (the first table in the list of the Title Block tables is considered to be the main, see Fig.10.2.5 on p. 1966).
Number of Rows in 1st Block	A field to input the number revision records in the first block of the table. E.g., the number of rows in the first block of the table should not exceed 4, so that the Revision Table does not go beyond the Title Block table limits on the first sheet of a design drawing.
Number of Rows in Next Blocks	A field to input the number of revision records in the subsequent blocks of the table. E.g., the number of rows in the subsequent blocks of Revision Table should be equal to 10 for the subsequent blocks of Revision Table to be equal by height to the Title Block table on the first sheet of a design drawing.
Place Next Block to the Right of Previous One	An option, which enabling specifies that a subsequent revision block will be placed to the right of the previous one. If this option is not enabled, the subsequent revision blocks will be placed to the left of the previous ones.

* You can work with the revision table in the document using this layout if the work with BOM is enabled.

The text document Layout Setup dialog (Fig.10.2.12) additionally includes the group of elements **Text Fields**. The distances between the internal border sides and the corresponding input field boundaries can be defined in this group of elements. Offset and text 1-st line indent values are measured from the input field boundary (see Fig.10.1.40 on p. 1948).

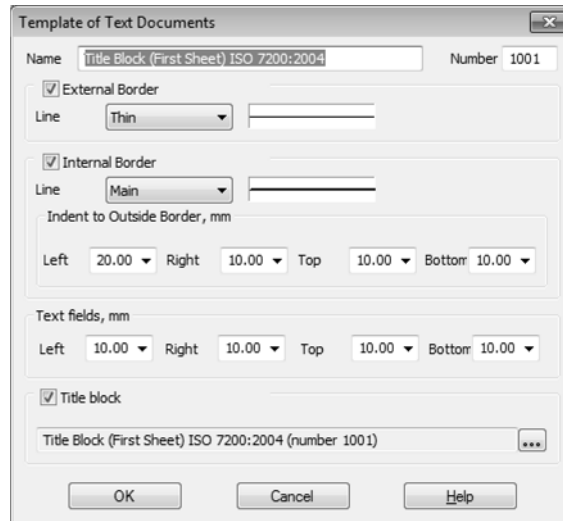


Fig. 10.2.12. Text Document Layout Setup dialog

10.2.1.8. Document Templates

KOMPAS-documents Template is the document blank containing several typical settings. Template files have the following extensions:

- ▼ Drawing Template—*cdt*,
- ▼ Fragment Template—*frt*,
- ▼ Text Document Template —*kdt*,
- ▼ BOM Template—*spt*,
- ▼ Part Template—*m3t*,
- ▼ Assembly Template—*a3t*.

The Template is selected when creating a document (Fig.10.2.13).

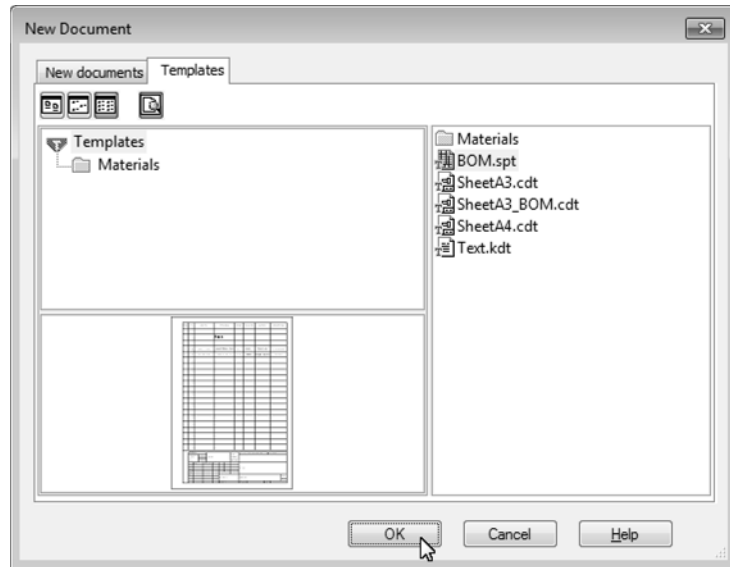


Fig. 10.2.13. Drawing Creation Using Template

Thanks to the use of Templates the process of preparation and setting up of new typical documents is expedited and unified.

The KOMPAS-3D package includes several Templates for drawings, text documents, BOM's and parts. The Templates are stored in subfolder *\Templates* of the main system folder.

User can edit the existing Templates and create the own ones for documents of any type.

The following steps should be carried out to make a Template.

1. Create a document of the required type.
2. Arrange and set it up as required.
3. Save the document by selecting row **Template** in list **Type** of the save dialog file.



Path to the templates folder (i.e. to the folder which is suggested by default for template saving) is defined by the `TEMPLATES` variable of the KOMPAS-3D environment (see Section 9.5.4 on p. 1886). By default, this folder is *\Templates* in the main system installation folder. If you need the path to the templates folder to be different from the default one, use the *Kompas.ini* file (see Section 9.5.3 on p. 1883).

The following settings and objects can be recommended to store in a Template:

- ▼ Format,
- ▼ Layout,
- ▼ Filling in the title block (*Name of Resp. Eng.*, *Name of Contr. Eng.*, *Name of Qual. Eng.*, *Company Name* etc.),
- ▼ Object settings (dimensions, leaders, etc.)
- ▼ System text styles settings,

- ▼ Property settings, part and assembly display accuracy.

When working with a document created using a Template, User can change any of the parameters stored in the Template.

10.2.2. Layout Creation practice

This Section includes some exercises, which will allow you to acquire the basic skills of work with the drawing and text document Layouts. Prior to proceeding to the exercises it is recommended to study carefully theoretic information described in the previous Section.

10.2.2.1. Techniques of Operation with Layout Libraries

The exercises to set up the Title Blocks and Layouts are recommended to be performed in a separate library created specially for training.

Exercise 17. Layout Library Creation. Independent work

Task. Create layout library `user_forms.lyt`.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Drawing and BOM Layout...**
2. Create a layout library with the name `user_forms` in a similar way as you created a line style library in Exercise 1 on p. 1924.

Proceed to the next exercise without closing the Layout Use dialog.

Exercise 18. Layout Copying between Libraries

Task. Copy the following system Layouts into library `user_forms.lyt`: **Design drawing. First Sheet. GOST 2.104–2006** or **Design Drawing. Final pages. GOST 2.104–2006** from library `Graphic.lyt`.

1. Activate the adjacent (to the left) preview window in the Layout Use dialog by clicking the mouse button.



2. Click button **Show Library**.
3. Open folder `\Sys` in the appeared dialog, select file `Graphic.lyt` and click button **Open**.

The list of Layouts stored in the selected library will appear in the preview window to the left.

4. Select the following Layouts in the list: *Design drawing. First Sheet. GOST 2.104-2006* or *Design Drawing. Final pages. GOST 2.104-2006*. To do this, select them simultaneously clicking the button `<Ctrl>`.



5. Click button **Copy**.

The selected Layouts will appear in a window to the right: the list of Layouts stored in library `user_forms.lyt`. These Layouts will be used to create custom Layouts when performing Exercises 22 and 23.

6. Close the drawing Layout Use dialog by clicking button **Exit**.

While copying layouts *Design drawing. First Sheet. GOST 2.104-2006* or *Design Drawing. Final pages. GOST 2.104-2006* title blocks that are used in these layouts were also copied to the *user_forms.lyt* library.

- In order to check is, click command **Service— Libraries of Styles, Layout Types and Title Blocks— Title Blocks...**

The Title Block Use dialog will appear on the screen. Note that the content of the same libraries as in the Layout Use dialog is displayed in preview windows of this dialog.

- Make sure that the preview window of library *user_forms.lyt* displays two Title Blocks *Design drawing. First Sheet. GOST 2.104-2006* or *Design Drawing. Final pages. GOST 2.104-2006*.

These Title Blocks will be used to create custom Title Blocks when performing Exercises 20 and 21.

- Close the Title Block Use dialog by clicking button **Exit**.

10.2.2.2. Drawing Layouts

This Section includes the exercises allowing to acquire skills of the Title Block and Layout creation. The use of creation methods is shown by the example of Layout creation for the first and next drawing sheets containing casting elements. In accordance with *Attachment 2 to GOST3.1125–88* these drawings should have an additional Title Block (Fig.10.2.14, 10.2.15). The purpose of the additional Title Block columns is described in Table10.2.8.

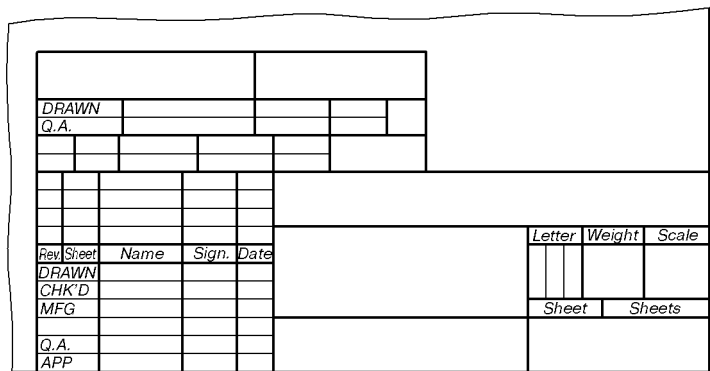


Fig. 10.2.14. Additional Title Block Location on Sheet

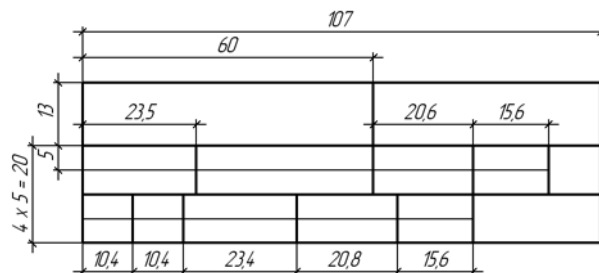


Fig. 10.2.15. Additional Title Block Structure and Dimensions

Tab. 10.2.8. Purpose of Additional Title Block Columns for Drawings Containing Casting Elements

Column number	Purpose
1	Casting code (or designation)
2	Document designation as per GOST3.1201–85.
3	Casting Resp. Eng.
4	Name of Casting Resp. Eng.
5	Signature of Casting Resp. Eng.
6	Signature date.
7	Qual. Eng.
8	Name of Qual. Eng.
9	Signature of Qual. Eng.
10	Document revision number.
11	Mark about notice list supersedence and introduction as per GOST2.503-90.
12	Notice designation (code).
13	Signature of person resp. for revision.
14	Spare.

10.2.2.2.1. Title Blocks

Exercise 19. Table Image Creation in Fragment

1. Create a new fragment.
2. Using system line styles *Normal* and *Thin* draw a table in this fragment following the dimensions specified on Fig. 10.2.15.

As it can be seen from Fig. 10.2.14, the lower border of additional Title Block coincides with the upper border of the drawing main table. Therefore, do not draw the line limiting the Title Block from below.

3. Save the created fragment under the name *stamp.frw* and close it.

Exercise 20. Title Block for First Sheet

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Title Blocks...**

The Title Block Use dialog will appear on the screen. If library *user_forms.lyt* is closed, open it using button **Show Library**.

When making test exercise18, we copied to library *user_forms.lyt* the layouts *Design Drawing. First Sheet. GOST 2.104-2006* or *Design Drawing. Final pages. GOST 2.104-2006*. As you re-



member, the respective title blocks were also automatically copied. Their names are displayed in the current preview window.



2. Select Title Block Text *Design Drawing. First Sheet. GOST 2.104-2006* and click the button **Edit**. The Title Block creation and editing dialog displaying parameters of the selected Title Block will appear on the screen. These parameters should be edited.
3. Enter value *101* into field **Number**.
4. Enter the following to the field **Name**: *Drawing with casting elements. First Sheet. GOST 3.1125-88*.
5. Click button **Add...**
6. Click button **From File** in the appeared table creation dialog box and select from menu command **Create from Fragment**.
7. Select *filestamp.frw*, which contains the table image, in the appeared file opening dialog.
8. Click button **OK** in the appeared table creation parameters dialog.
9. The table, the image of which was stored in the selected fragment, will appear in KOMPAS-3D window (Fig. 10.2.16).

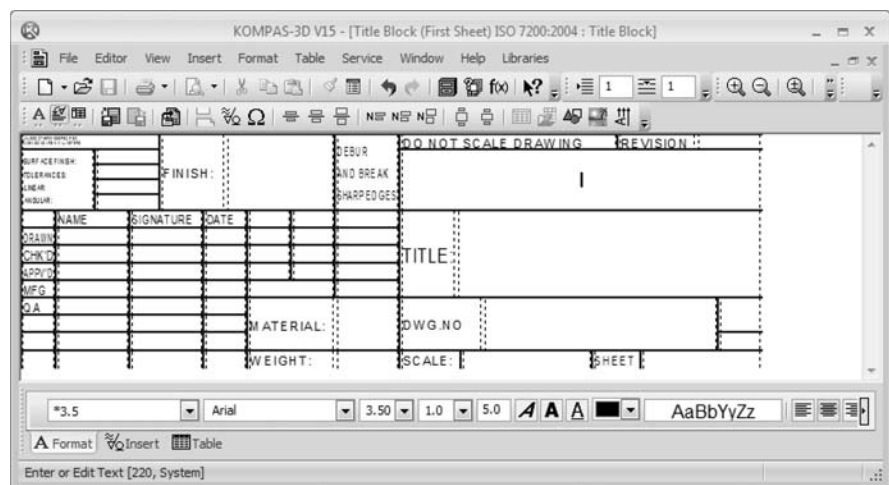


Fig. 10.2.16. KOMPAS-3D window in mode of new title block creation



As this fragment did not include the line limiting the Title Block from below, then style *Hidden* was automatically selected for lower border lines of the corresponding table cells. By default, hidden lines are not displayed on the screen. To display them click button **Show Grid** on page **Table** of the Property Bar.

The text input boundaries are shown with dashed line inside the cells. Distances from input area boundaries to the corresponding cell boundaries are text **Offsets**. The following offset values are used by default:

- ▼ left 0.5 mm
- ▼ right 0.5 mm
- ▼ top 0 mm

▼ bottom 0 mm

10. Set top and bottom offsets equal to 0.5mm for columns 1, 2 and 14.
 - 10.1. Place the cursor into the required cell and call command **Format— Cell Format**.
 - 10.2. Enter value **0.5** in fields **Top** and *Bottom* of the appeared dialog box.
 - 10.3. Left unchanged the other offset values of these and other cells.
11. Call command **Table— Table Fixing**.
12. Enable all options in the appeared dialog box and close it.
13. Set text style *Default* to all columns, except for columns 1, 2, 3, 7 and 14, and enable option to enter only one text line to each of them.
 - 13.1. Place the cursor into the required cell and call command **Format— Cell Format**.
 - 13.2. Enable option **Single-line Text** in the appeared dialog box.

Owing to this setting the line break will be ignored in these columns when filling in the additional Title Block. The string length will be fitted to the column width automatically increasing text symbol compression.
 - 13.3. Click button **Change...**
 - 13.4. Activate the **Embedded** tab in the displayed dialog.
 - 13.5. Select style *Default* from the list on this page and click button **Select**.

The name of the selected style will appear in field **Default Text Style** of the cell format setup dialog.
 - 13.6. Close the cell format setup dialog.

How did text styles *Default*, *Letter-Mass-Scale*, *Designation*, *Name* and *Material-Company*, the names of which are displayed in the current text style selection dialog, appear in a new table of the Title Block?

They were created and saved in Title Block *Design Drawing. First Sheet. GOST 2.104-2006*, which was used as the basis for a new Title Block.

Text style creation in the Title Block is possible when creating or editing any table of this Title Block. Text styles created in the Title Block are available in any of its tables, but not only in that one, for which they were created.

You just checked it by assigning style *Default* to the cells of a newly created table. This style will be applied to the texts entered to these cells when filling in the additional Title Block of a document.
14. Assign text style *Default* to columns 3, 7 and 14.

It is not necessary to set a ban for line break in these columns. Columns 3 and 7 will contain text and as a consequence they will not be available to users. Column 14— a spare one, so a possibility of multiple line input should be left for it.
15. Set left alignment for all columns, except for columns 1 and 2.
16. Enter text *Name of Resp. Eng.* into column 3, and text *Name of Qual.Eng.* into column 7.

Text input and edit will be impossible in these columns while filling in the additional Title Block.
17. Assign style *Designation* to columns 1 and 2.

This style has the advanced style setting— a setting determining line spacing and symbol size and spacing in the text depending on the number of lines in it.

17.1. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Text Styles...** to see this advanced style setting.



17.2. Click button **Show document** in the appeared dialog box.

17.3. Activate the **Embedded** tab in the active preview window. The list of text styles available in the current Title Block (the Title Block, which includes the table to be edited) is displayed on this page.

17.4. Select style *Designation* from the list.



17.5. Click button **Edit**.

17.6. In the appearing dialog, click the **Next...** button.

The text advanced style setup dialog will appear on the screen (Fig.10.2.17).

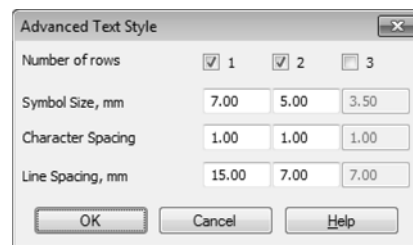


Fig. 10.2.17. Text Advanced Style Setup Dialog

The settings in this dialog determine the following rules of text location in the cell:

- ▼ if the text consists of one line, then symbol size will be 7mm; line spacing is of no importance in this case,
- ▼ if the text consists of two lines, then they will be located with spacing 7mm; symbol size is 5mm.

17.7. Without changing the settings close the dialogs of text advanced style setup, text style edit and use of style library sets.

General cell setting up— determining of text appearance in the cells— is finished. Now, it is necessary to set up the advanced format— determine service functions available when filling in the additional Title Block columns in a document, as well as the possibility and order of data transfer from one additional Title Block to another one when changing the document Layout.

The advanced format settings should be made for each empty cell, i.e. for all cells, except for 3 and 7. For this purpose, the cursor should be positioned in the required cells one by one, the advanced format setup dialog should be called (Fig. 10.2.18), and the required parameters should be set.

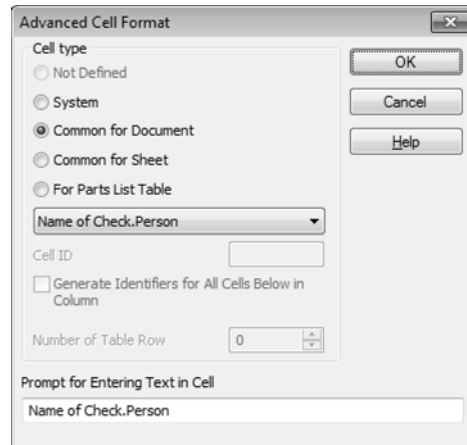


Fig. 10.2.18. Advanced Cell Format Settings dialog



18. Advanced Cell Format Settings dialog. To do this, click button **Advanced Cell Format** on toolbar **Tables and Borders**.

19. Set up the advanced cell format in accordance with Table 10.2.9.

Tab. 10.2.9. Advanced Cell Format Setup

No. Column No.	Cell Type	Cell Identifier	Value (for identifier Custom)
1	Common for Document	Custom	1000
2	Common for Document	Custom	1001
4	Common for Document	Custom	1002
5	Common for Sheet	1003	
6	Common for Document	Custom	1004
8	Common for Document	Name of Qual. Eng.	
9	Common for Sheet	1005	
6A	Common for Document	Qual. insp. date	

20. Set type **Common for Sheet** for all other cells and assign to them identifiers from 1006 to 1017.

21. Enter information about the column purposes from Table 10.2.8 on p. 1986 as **Prompt for Entering Text in Cell**.

So, e.g., *Casting code or designation* will serve as a prompt for the first column, and *Document designation as per GOST 3.1201–85* for the second column, etc.

22. Enter vertical line and number 4099 after the prompt text when setting up the columns for date input (6, 6a, 6b and 6c): 4099

Owing to this fact, the double mouse click in these columns will call in future (when filling in the additional Title Block) the date selection dialog.

Why should the table cell advanced format be set up in this particular way?

- ▼ None of the cells contains the data, which could be determined by the system independently, so none of the cells has type **System**.
- ▼ In just the same way none of the cells is intended for BOM object input, so type **For BOM Table** is not used either.
- ▼ Following the rules given in Sections *Peculiarities of Using Cells of Types Common for Sheet and Common for Document* on p. 1973 and *Use of Custom Identifiers* on p. 1974, the cells of a created table should be set up as follows.
 - ▼ Casting code and document designation is information, which should be the same on all sheets of a multisheet document, so type **Common for document** is selected for columns 1 and 2. Identifier **Custom** is selected for these cells as there is no system identifier for them. Identifier values are set in accordance with Table 10.2.4 on p. 1975.
 - ▼ Name of Resp. Eng. of the casting drawing is the same on all sheets of a multisheet drawing. The same can be said about the Name of Qual. Eng. That's why type **Common for Document** is selected for columns 4 and 8.
 - ▼ System identifier *Name of Resp. Eng.* can not be assigned to column **Casting Resp. Eng.** as usually a casting drawing responsible engineer and a part drawing responsible engineer (whose name is specified in the Title Block main table) are two different persons. It would have led to transferring the Name of Resp. Eng. from the Title Block to this column. Therefore, the custom identifier is assigned to column 4.
 - ▼ Contrary to the responsible engineers, the quality engineer for a casting and part drawing can be one and the same person, so the following system identifier is assigned to column 8: **Name of Qual.** Therefore, the name entered into column *Name of Qual. Eng.* while filling in the Title Block will be automatically transferred to column *Name of Qual.*
 - ▼ Columns 5, 9, 13 and 13a are provided for signatures, therefore they have type **Common for Sheet** and unique numeric identifiers.
 - ▼ Columns 6 and 6a are provided for dates of document signing by the casting responsible engineer and the quality engineer. These columns are set up analogous to columns 4 and 8: both of them have type **Common for Document**, but Custom identifier is assigned to column 6 and System identifier is assigned to column 6A (**Qual. Insp. Date**).



If quality engineers, which check part drawings and casting drawings, are different, then columns 8 and 6a should be set up analogous to columns 4 and 6.

- ▼ Columns 10, 10a, 11, 11a, 12 and 12a include the data unique for this sheet, therefore they have type — **Common for Sheet**.
23. Call command **Service— Identifier Check**.

The dialog with the current table image will appear on the screen. Displaying of any table cells in blue or pink color denotes correspondingly the absence or the coincidence of identifiers of these cells.

24. Provided that such cells are present in the table, check and change their settings.
25. Call the **File— Save Table** command.
The created table and its settings will be recorded into Title Block *Drawing with casting elements. First Sheet. GOST 3.1125–88*.
26. Now save table to file. This file will be used when creating the Title Block for next sheets of drawings with casting views.
 - 26.1. Call the **File— Save Table to File...** command.
 - 26.2. Select a folder to save the table file in the appeared dialog box. Enter **stamp** into field *File Name* and click button **Save**.
File *stamp.tbl* containing structure and Layout of this table, format and advanced format of its cells will be created in the specified folder.
27. Call the **File — Finish table editing** command or click the edit mode symbol in the table window.
Table editing window will be closed. The Title Block creation dialog will appear again on the screen.
A newly created table will be placed to the end of list **Components of Title Block** and will be called “UNNAMED”. Its image will be highlighted with green in window **Print Preview**.
28. Select this new table in the list and enter *Additional* into field **Table Name**.
29. Locate this new table on sheet in such a way that its lower left corner coincides with the upper left corner of the Main Table.
 - 29.1. Select line **Right Bottom Corner of Border** from list **Base Point on Sheet**.
 - 29.2. Select line **Left Bottom Corner of Border** from list **Table Base Point**.
 - 29.3. Enter value **-185** into field *Horizontal Offset*.
 - 29.4. Enter value **55** into field *Vertical Offset*.
 - 29.5. Enter value **0** into field *Rotation Angle*.
30. Do not enable options **Relative to longer side** and **Use for BOM**.
31. At that the process of creation of Title Block *Drawing with casting elements. First Sheet. GOST3.1125–88* is finished. Click **OK**.
The Title Block creation dialog will be closed. The Title Block Use dialog will appear on the screen. It contains title blocks *Design drawing. Final pages. GOST 2.104-2006* and the *Drawing with casting elements. First Sheet. GOST 3.1125–88*.

Exercise 21. Title Block for the Next Sheets

The creation process of the Title Block for next sheets does not include the step of table setting up, as the table for this Title Block is ready already— it is stored in file *stamp.tbl*. Other steps are analogous to those of the Title Block creation for the first sheet, so their detailed description is omitted.



1. Select Title Block Text *Design Drawing. Final pages. GOST 2.104-2006* and click the button **Edit**.
The Title Block creation and editing dialog shall appear on the screen.
2. Enter **102** in the field *Number*.

3. Enter the following to the field **Name:** *Drawing with castings elements. Final pages. GOST 3.1125–88.*
4. Click button **Add...**
5. In the displayed table creation dialog, click button **From File** and select from menu the command **Load from Table File...**
6. In the appeared file opening dialog, select file *stamp.tbl*, which contains the table created and set up when performing the previous Exercise.
The selected table will appear in KOMPAS-3D window.
7. Make sure that the table retained its structure and layout and its cells retained their texts, format and advanced format settings.
8. Call the **File— Save Table** command.
9. Repeat items 28–30 from the previous exercise by setting the vertical offset to 15mm.
10. At that the process of creation of Title Block *Drawing with casting elements. Final pages. GOST 3.1125–88* is finished. Click **OK**.
The Title Block creation dialog will be closed. The Title Block Use dialog will appear on the screen. Now it includes the following Title Blocks: *Drawing with casting elements. First Sheet. GOST 2.104-88* and the *Drawing with casting elements. Final pages. GOST 3.1125–88.*
11. Close the Title Block Use dialog.

10.2.2.2.2. Templates

Exercise 22. First Sheet Layout

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Drawing and BOM Layout...**

The Layout Use dialog will appear on the screen. The list of Layouts stored in library *user_forms.lyt* is displayed in one of its windows: *Design drawing. First Sheet. GOST 2.104-2006* or *Design Drawing. Final pages. GOST 2.104-2006.*



2. If this is not the case, click button **Show Library** and open file *user_forms.lyt*.
3. Select Layout *Design drawing. First Sheet. GOST 2.104-2006* and click the button *Edit*.



The document Layout Setup dialog will appear on the screen.

4. Enter the following to the field **Name:** *Drawing with castings elements. First Sheet. GOST 3.1125–88.*
5. Enter **101** in the field *Number*.
6. Click button **Select** in group **Title Block**.
7. Select the following line in the appeared Title Block Selection dialog: *Drawing with castings elements. First Sheet. GOST 3.1125–88* and click the button *OK*.
The Title Block Selection dialog will be closed. The Layout Setup dialog will remain on the screen.
8. Close the Layout Setup dialog without changing other parameters by clicking **OK**.

The Layout Use dialog will remain on the screen. Instead of *Layout Design drawing. First Sheet. GOST 2.104-2006* the following Layout will appear in its active preview window: *Drawing with castings elements. First Sheet. GOST3.1125-88*.

Exercise 23. Layout of Next Sheets

1. Select *Layout Design drawing. Final pages. GOST 2.104-2006* and click the button **Edit**.
The document Layout Setup dialog will appear on the screen.
2. Enter the following to the field **Name**: *Drawing with castings elements. Final pages. GOST 3.1125-88*.
3. Enter **102** in the field *Number*.
4. Select the *Drawing with casting elements* title block. *Final pages. GOST 3.1125-88*.
5. Do not change other parameters.
6. Click **OK**.
7. Close the Layout setting dialog.

10.2.2.2.3. Layout Correctness Check

It is necessary to check the Layout setting correctness immediately after their creation. Having found an error, correct it immediately because edition of the Layouts already assigned to documents is a very long and laborious work.

Exercise 24. Check of Created Layouts

1. Create a new drawing in KOMPAS-3D.
The following default Layout is assigned to the first sheet of a new drawing *Design drawing. First Sheet. GOST 2.104-2006* from library *Graphic.lyt*.
2. Assign to this sheet the Layout *Drawing with castings elements. First Sheet. GOST3.1125-88* from library *user_forms.lyt*.



- 2.1. Call command **Service— Document manager**.
- 2.2. Look at the Tree of sheets, views and layers (it is located in the left part of dialog **Document manager**) and select object **Sheets** in it.



The line corresponding to a single sheet will appear in the right part of the dialog — in the List of Sheets, Views and Layers. List of Sheets, Views and Layers is a table with object properties displayed in its columns. Any property can be changed directly in Document Manager.

- 2.3. Select the line corresponding to the drawing first sheet by a mouse click.
- 2.4. Click on the last cell of the line— a cell located in column **Layout Library**.

The sheet Layout Selection dialog will appear on the screen.



- 2.5. Click button **Library** in it.
- 2.6. Select file *user_forms.lyt* in the appeared file opening dialog and click button **Open**.
The file opening dialog will be closed. The dialog of Layout Selection from the Library will appear on the screen.
- 2.7. Select line *Drawing with casting elements. First Sheet. GOST 3.1125-88*.

- 2.8. Close the dialog of layout selection from the library using **OK** button and then click the **Re-read** button in the dialog for page layout selection.
- 2.9. Click button **Apply** in dialog **Document Manager**.
You will see that the drawing sheet image has changed on the screen: now it includes the additional Title Block. Move window Document Manager sideways not closing it for convenient view.
3. Add a new sheet to the drawing and assign the Layout *Drawing with castings elements. Final pages. GOST 3.1125–88*. These operations are executed via Document manager.
 - 3.1. Chose a convenient screen position for the Document manager to facilitate your work.
 - 3.2. Click the Sheet creation button on **Document Manager** toolbar.
The line corresponding to the added sheet shall appear in the List of Sheets, Views and Layers. By default, the added sheet has the *Design drawing* layout. *Final pages. GOST 2.104-2006* from library *Graphic.lyt*.
 - 3.3. Assign to this sheet the Layout *Drawing with castings elements. Final pages. GOST3.1125–88* from library *user_forms.lyt*. To do this, complete steps described in paragraphs 2.3–2.8.
4. Click **OK** to close the **Document Manager** dialog.
5. Check correct setup of the additional Title Block table.
 - 5.1. Click on the Title Block of the first sheet to enter this Block edit mode.
 - 5.2. Set cursor to column 1— the upper left cell of the table.
 - 5.3. Make sure that the Message line displays message “Casting Code or Designation”.
 - 5.4. Enter several symbols to the cell. Make sure that height of the symbols is not more than 7mm.
 - 5.5. Press the *<Enter>* key to create another line in the same cell, and enter the text of this line. Make sure that height of the symbols is 5mm.
 - 5.6. Double mouse click on any of the columns provided for date input (6, 6a, 6b or 6c). Make sure that the Date Selection dialog appeared on the screen.
 - 5.7. Fill in all columns of the additional Title Block with the arbitrary data, except for the columns provided for signatures (5, 9, 13 and 13a).
Make sure that arbitrary number of lines can be entered into column 14 and only one line—into all other columns.
Make sure that style *Default* is applied to the entered text.
 - 5.8. Exit from the Title Block edit mode by pressing key combination *<Ctrl> + <Enter>*.
Make sure that:
 - ▼ the data entered into columns *Casting code or designation* and *Document designation as per GOST 3.1201–85* (1 and 2) have been transferred to the corresponding cells of the additional Title Block on the second sheet;
 - ▼ the data entered into columns *Name of Casting Resp. Eng., Name of Qual.Eng., Resp. Eng. signature Date, Qual. Eng. Signature Date* (4, 8, 6 and 6a) have been transferred to the corresponding cells of the additional Title Block on the second sheet;



- ▼ the data entered into columns *Name of Qual.Eng.* and *Qual. Eng. Signature Date* (8 and 6a) have also been transferred to the corresponding columns of the Title Block Main Table on the first sheet;
 - ▼ the data about casting modifications have not been transferred anywhere.
6. Save your drawing.
 7. Assign to the first sheet the Layout Drawing with castings elements. Final pages. GOST3.1125–88.

Make sure that all the data remained unchanged in the additional Title Block after the Layout change.



Actually, the following had happened when the Layout was changed. Title block tables included in the *Drawing with castings elements layouts. First sheet. GOST 2.104-88* and the *Drawing with casting elements. Final pages. GOST 3.1125–88* were checked for identical cells having similar types and identifiers. Then, the data from cells of the previous Title Block were transferred into identical cells of a new Title Block.

In this case the identical cells were found owing to the use of the same table in Layouts.

If it is required to organize the data transfer between the cells of different tables included into different Title Blocks, it is necessary to set up these cells manually in such a way that their types and identifiers coincide.

10.2.2.3. Layout of Text Documents

This Section includes Exercises, which allow to get acquainted with some peculiarities of the text document Layouts.

Layout creation for text documents includes the same two main steps as for drawing Layout creation:

- ▼ Title Block creation,
- ▼ creation of the Layout containing the Title Block.

The process of text document Title Block creation is analogous to drawing Title Block creation.

The Layout creation using a ready Title Block is the subject of Exercise 25.

Exercise 26 is dedicated to creating additional sheets in the text documents.



The Layout, the creation process of which is described in Exercise 25, already exists in library *Graphic.lyt*.

Exercise 25. Layout for Text Document

Task. Create a Layout in library user_ forms for the revision sheet in a text document. Use **Title Block Revision** sheet (vertic.) (vertic.) **Final pages. GOST 2.503–90 FZ** from library *Graphic.lyt*.

1. Call command **Service — Libraries of Styles, Layout Types and Title Blocks — Title Blocks...**

The Title Block Use dialog will appear on the screen.



2. Using button **Show library**, open library *user_forms.lyt* in one window of the dialog and library *Graphic.lyt* in the other one.
3. Copy the *Title Block Revision sheet. (vertic.) Final pages. GOST 2.503-90 F3* from library *Graphic.lyt* to *user_forms.lyt* and close the Title Block Use dialog.
4. Call command **Service— Libraries of Styles, Layout Types and Title Blocks— Layout of text documents...**

The text document Layout Use dialog will appear on the screen.

5. Open library **user_forms.lyt** using button *Show library*.
6. Create *Layout Revision sheet (vertic.) Final pages. GOST 2.503-90 F3*



- 6.1. Click button **Create** in the text document Layout Use dialog.

The text document Layout Setup dialog will appear on the screen.

- 6.2. Enter the following to the **Name** field: *Revision sheet. (vertic.) Final pages. GOST 2.503-90 F3*
- 6.3. Enter **101** in the field *Number*.
- 6.4. Enable option **Title Block**. A field with the Title Block name is located under it. For now this field is empty.
- 6.5. Click button **Select** to the left from the field with the Title Block name.
- 6.6. The Title Block Selection dialog will appear on the screen.
- 6.7. Select the following Layout in this dialog: *Revision sheet. (vertic.) Final pages. GOST 2.503-90 F3* and click the **OK** button.

The Title Block Selection dialog will be closed, and the name of the selected Title Block will appear in the text document Layout Setup dialog.

At that the Layout creation process is finished (Fig.10.2.19).

Fig. 10.2.19. Template Creation for Revision Sheet in Text Document

- 6.8. Close the dialog without changing other parameters by clicking **OK** button.

The text document Layout Use dialog will remain on the screen. The following newly created Layout will appear in the list of text document Layouts stored in library *user_forms.lyt:Revision sheet. (vertic.) (vertic.) Final pages. GOST 2.503-90 F3*

7. Close the text document Layout Use dialog.

Exercise 26. Inclusion of Additional Sheets into Text Document

Task. Create a text document with a cover sheet at the beginning and a revision sheet at the end of this document.

1. Create a new text document in KOMPAS-3D.
The document is displayed in normal mode, i.e., without border and Title Block.
2. Fill in 2-3 pages of the document with any text.
3. Call command **View— Page Layout**.
The document image will change: border and Title Block will appear on the screen. Make sure that the document does not include any additional sheets.
4. Create additional sheets in the current document.
 - 4.1. Call command **Service — Parameters... — Current Text Document — Sheet Parameters — Additional sheets**.
The additional sheets Setup dialog for the current text document will appear on the screen.
 - 4.2. Click button **Add** in group **At Document's Beginning**. The Layout Selection dialog will appear on the screen.
 - 4.3. Click button with three dots to the right from list **Name**.
The dialog of Layout Selection from the Library will appear on the screen.
 - 4.4. Select line **Cover sheet. GOST2,105-95**.
 - 4.5. Close the dialog of Layout Selection from the Library and then the dialog **Layout** by clicking the **OK** button.
The additional sheets setup dialog will remain on the screen. The name of the selected layout will appear in the list of additional sheets **At Document Beginning— Cover Sheet. GOST2,105-95**.
 - 4.6. In a similar way add Layout at the end of the document a sheet with **Layout Revision sheet (vertic.) Final pages. GOST 2.503-90 F3**, which was created when performing the previous Exercise.
 - 4.7. At that the setup process for additional sheets of the current text document is finished. Close the dialog by clicking the **OK** button.
5. Make sure that a cover sheet appeared at the beginning of the document (before its first sheet) and revision sheet— at the end (after its last sheet).



Text input on these sheets is carried out in the Title Block editing mode.

6. Make sure that both additional sheets are included into automatic sheet numbering and automatic total sheet number determination, i.e., actually, the first additional sheet at the beginning

should become the first sheet of the document and the last additional sheet at the end should become the last sheet of the document.

**11. Import and Export.
Hyperlinks.
Team work.
Document recovery**

11.1. Import and export

11.1.1. Import and Export of Graphical Documents

The exchange of graphic documents between KOMPAS-3D and other systems is possible through the formats listed in the Table 11.1.1. Numbers in the “Import” and “Export” columns denote the number of the format version, with import being available for both the indicated version of the format and all its previous versions, while export is only possible to the specified format version(s).

Tab. 11.1.1. Graphics exchange formats

Format	Import	Export
IGES (*.igs)	5.3	5.3
AutoCAD DXF (*.dxf), AutoCAD DWG (*.dwg)	2013	2000, 2004, 2007, 2010, 2013 (version selected by user during export setup)

Reading of txt and rtf text documents is also possible.

You can also save graphic and text documents KOMPAS-3D to:

- ▼ previous format versions of KOMPAS-3D: KOMPAS 5.11 R03 and KOMPAS-3D V13,
- ▼ bitmap formats (GIF, JPEG, etc.),
- ▼ Windows Metafile (*.wmf) and Enhanced Metafile (*.emf),
- ▼ Portable Document Format (*.pdf).
- ▼ txt (only for text documents).



Export of a document results in creating a new document of the selected format, visually identical to the source one. However, the structure and internal connections of objects in the KOMPAS-3D documents and of the same objects in the new format may differ. Dimensions, extended lines etc. are typical examples of such objects. When editing the target document, the appearance of such objects may change so that they lose identity to objects of the source document. Similar changes of objects may take place under multiple conversion of a document from the KOMPAS to other formats, and visa versa.

11.1.1.1. Import

To read document of any of the above mentioned formats, do the following.

1. Call the **File — Open** command.
2. In the **File type** list: of the opened dialog, select the desired format and specify the file name.
3. Click the **Open** button. The selected document will be imported to KOMPAS-3D and loaded for editing.

There are certain peculiarities of graphic document import from files of some formats.

If a document is imported from a file of IGES format, once the **Open** button is clicked, a dialog will appear where the type of the file being opened should be specified (2D or 3D). After that, the import setup dialog will be displayed. Once you set import parameters, click the **Start Reading** button.

When importing documents from a file of DXF or DWG format, when a selection is made from the **File type list**: If you select **AutoCAD DXF (*.dxf)** or **AutoCAD DWG (*.dwg)**, the following controls will appear in the Open File dialog.

- ▼ Options **Read to Graphical Document** and **Read to Model**, which allow you to specify the type of the document to be opened. To read either DXF or DWG drawing, enable option **Read in Graphical Document**.
- ▼ Arrow in the right part of the **Open** button, which serves to expand this button's menu consisting of **Open** and **Open with Parameters...** commands. To call the import setup dialog, select the **Open with Parameters...** command. Once you have set the import parameters, click the **Start Reading** button in the setup dialog.

11.1.1.2. Export

To save KOMPAS document in other formats, do the following.

1. Call the command **File—Save As...** The file save dialog will appear. Depending on the current document type, the **File Type** list of this dialogue will contain various names of the formats available for saving.
2. Select the desired format and click the **Save** button.

When a selection is made in the **File type list**: rows **AutoCAD DXF (*.dxf)** and **AutoCAD DWG (*.dwg)** the **Parameters** button will appear in the dialog. It serves to call the dialog to set up document export in the selected format. After finishing the setup close the dialog by clicking the **OK** button, and then click the **Save** button of the file saving dialog. The selected document will be saved to a file of the specified format.

You cannot set up import for **IGES** format.

Setup for saving documents in the **WMF** and **EMF** formats is performed using the dialog that appears automatically upon clicking the **Save** button. This dialog is similar to the dialog, which is used to save bitmapped files, the only difference is that it has no elements to control bit map parameters. Setup dialogs, used to save documents of various types in bit map formats, are described in Section 11.1.3.

Export to **IGES** and **WMF** formats is limited. See the following Table 11.1.2.

Tab. 11.1.2. Saving limitations for KOMPAS documents in IGES and WMF formats

Format	Saving limitations
IGES	Inserts of views, multilayer fragments, and multilayer macro elements are not saved.
WMF	Color fills, bitmaps, and OLE-inserts are not saved.



A document saved in other format can be previewed and printed.

Export of a document results in creating a new document of the selected format, visually identical to the source one. However, the structure and internal connections of objects in the KOMPAS-3D documents and of the same objects in the new format may differ. The AutoCAD drawings layout elements are typical examples of such objects. When editing the target document, the appearance of such objects may change so that they lose identity to objects of the source document.

11.1.2. Import and Export of Models

Exchange of 3D-models between KOMPAS-3D and other systems is possible through the formats listed in the Table 11.1.3. Numbers in “Import” and “Export” columns mean the number of a format version, where import option applies to both the indicated version of the format and all its previous versions, whereas export is only possible into the indicated format version.

Tab. 11.1.3. Formats of 3D model exchange

Format	Import	Export
IGES (*.igs)	5.3	5.3
ACIS (*.sat)	8.0	2.0
Parasolid (*.x_t, *.x_b, *.xmt_txt, *.xmt_bin, *.xmp_txt, *.xmp_bin)	19.0	10.0 (export is possible into files of the *.x_t and *.x_b types)
STEP AP 203 (*.stp, *.step)	1.0	1.0
STEP AP214 (*.stp, *.step)	1.0	1.0
STL (*.stl)	Import is not available	format has no version control
AutoCAD DXF (*.dxf), AutoCAD DWG (*.dwg)	2007	export is not available

You can also save KOMPAS-3D V15 models to:

- ▼ previous format versions of KOMPAS-3D: KOMPAS 5.11 R03 and KOMPAS-3D V14,
- ▼ bitmap formats (GIF, JPEG, Windows Metafile, etc.),
- ▼ view format VRML (*.vrl); format version— 2.0,
- ▼ math core format C3D (*.c3d),
- ▼ Portable Document Format (*.pdf).

11.1.2.1. Import

To read the document in any one of the importable formats, perform the following operations.

1. Call the **File — Open** command.
2. In the **File type:** list of the dialog box, select the desired format and specify the file name.
3. If the DXF or DWG format is selected, the right-hand part of the dialog will have the options: **Import to Graphics Document** and **Import to Model**. You can use these options to specify the type of the opened document. Enable the **Import to Model** option for reading a model.
4. For some formats, the right-hand side of the **Open** button will have a downward arrow that makes it possible to open the dropdown menu. The menu consists of commands **Open** and **Open with parameters...**
 - ▼ The **Open** command allows importing a model without changing import parameters (calling this command is the same as clicking the **Open** button).
 - ▼ The **Open with parameters...** command may be used to set up the model import parameters. Upon calling the command, the import setup dialog will appear. Make necessary adjustments and click **OK** on the dialog box or the **Start Reading** button (for DXF or DWG formats).

The selected document will be imported to KOMPAS-3D and loaded for editing.

The part import operation generates one file — a KOMPAS-3D part displayed in the current window.

The assembly import operation generates several files:

- ▼ main assembly;
- ▼ main assembly components: subassemblies and parts.

The current window displays the main assembly.

Peculiarities of importing model objects (bodies, surfaces and so on) are described below.

11.1.2.1.1. Peculiarities of model object import

When importing a model from all importable formats, surfaces and bodies are transferred.

Surfaces are transformed into “imported surfaces”, while bodies are transformed into “operations without history”.

When importing a model from IGES and STEP AP203/STEP AP214 formats, curves and points are transmitted in addition to surfaces and bodies.

Points are transferred with retaining the object's type.

There are the following peculiarities of importing curves: segments, arcs, polylines, and splines are transferred with saving object types, other curves are transformed into “curves without history”.

When importing from the STEP AP214 format, dimensions and specifications are transmitted. Dimensions are converted into 'dimensions without history'.

The texts of technical parameters and dimensions are transmitted as letters, characters and special symbols or as sets of sections grouped in macroelements.

A special feature of reading dimensions with limit deviations is that symbols denoting deviations are placed in one row with the nominal dimension value and have the same height. The upper deviation is placed first, followed by the lower.

When importing models from the DXF or DWG format, curves (segments, arcs) are transferred apart from surfaces and bodies.

11.1.2.2. Export

Perform the following actions to save a KOMPAS model into another format.

1. Call the command **File** — **Save As...** The file save dialog will appear. Depending on the type of the current document, the **File type:** list of this dialog contains different format names that are available for saving.
2. Select the desired format and click the **Save** button.

The export set up dialog appears when you export files to specific formats. For the rest of formats, export cannot be customized.

Exporting the model into IGES, STEP AP203/STEP AP214 and STL formats has the following trait.

After selecting a string with one of the said formats from the **File type** list, the right-hand side of the **Save** button will have a downward arrow that makes it possible to open a dropdown menu. This menu consists of the following commands: **Save** and **Save with parameters...** In order to export a model without additional export parameters setup, select **Save**. The **Save with parameters...** command makes it possible to setup model export parameters. Upon calling the command, the export setup dialog will appear. After making the required settings, click the **OK** button.



If saving the model in another format is performed in recalculation mode, the model is saved with the dimensions of the current mode. For more information about the model dimension recalculation mode, see Section 2.10.3 on p. 671.

11.1.2.2.1. Peculiarities of model object export

When exporting a model to different formats, the following objects are transferred:

- ▼ bodies and surfaces export into the Parasolid, ACIS and STL formats,
- ▼ bodies, surfaces, points, curves export into STEP AP203 /STEP AP214 and IGES formats.

In addition to that, dimensions and specifications are transmitted to the STEP AP214 format and sketches are exported into IGES.

Bodies, surfaces, and points are transferred without modifications.

There are the following peculiarities of exporting curves.

- ▼ Segments, arcs, polylines, and splines are transferred without any modifications.
- ▼ Curves by law, isoparametric curves, spirals (cylindric and conic), rounding curves, and curve joints are transformed into splines.



If a rounding curve is the arc, it will be transferred without modifications.

- ▼ Equidistants are transformed into curves that are the same as the base ones, for example, if a segment is the base curve, the equidistant will be transformed into segment, if an arc – into an arc, if a curved segment of a curve – into a spline.

The dimensions are transferred as a set of 3D curves and texts.

11.1.3. Saving to bitmap and PDF formats

To save a document as a bitmap or PDF file, call the command **File — Save As...** In the dialog that appears, specify the folder and the file name. To save the document, click **Save**.

Save parameters can be set in a special dialog.

When you save a document to the bitmap format a save file parameters dialog appears automatically after you click **Save** in the file save dialog. When you save a document to PDF, the dialog can be brought up by clicking the **Parameters** button in the file selection dialog.

After you are done setting the parameters, click **OK**.

Sets of setup dialog control elements differ slightly depending on the saving document type (see Sections 11.1.3.1–11.1.3.3).

If, when saving to bitmap, no file is created, a message will appear explaining reasons for that. See procedure recommended in such cases and conditions, fulfillment of which is necessary for bitmap saving in the Section 11.1.3.4 on p. 2015.



Saving as bitmap or PDF is controlled by print filter settings, i.e. the result bitmap image will contain only those objects of the source document for which printing is enabled. See Section 9.1.5.2 on p. 1701 for the print filter settings.



When exporting a model to PDF, bodies and surfaces are exported along.

11.1.3.1. Drawing, Fragment, Model

Parameter setup dialog for saving a drawing, fragment, or model as a bitmap or PDF (graphics documents) file is shown in Fig. 11.1.1.

Control elements boxes are described in Table 11.1.4.

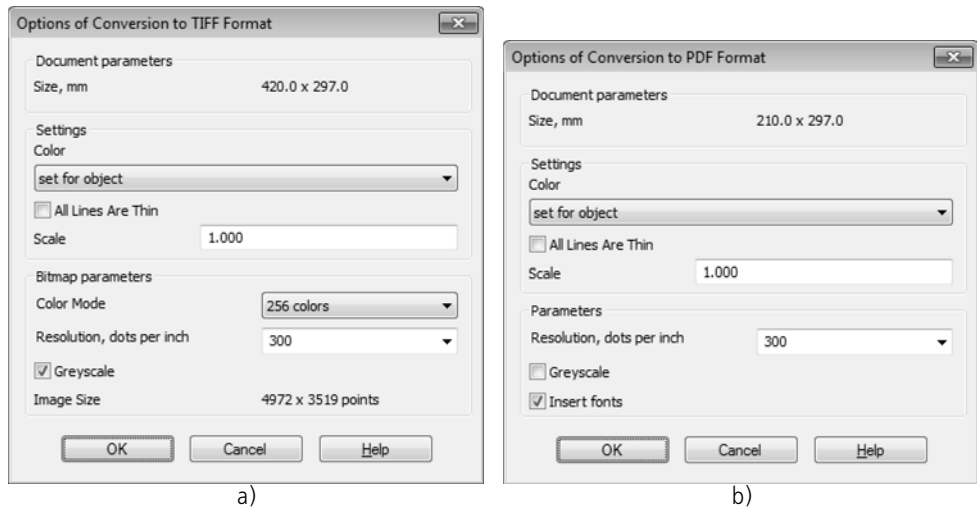


Fig. 11.1.1. Drawing or fragment saving setup dialog:
a) bitmap, b) PDF

Tab. 11.1.4. Saving setup dialog for bitmap and PDF

Element	Description
Parameters of a document	Reference information on saving document.
Color	You may choose the color of image from the list (see Table 11.1.5).
All Lines are Thin	This option allows saving all lines of 1 pixel width ^{***} .
Scale	This field serves for entering the image scale.
Color Mode*	The list serves to select the number of colors. Since the size of a bitmap file is proportional to the number of colors, we recommend using high color resolution only for recording three-dimensional models, as that will help render color transitions correctly. After saving the file, view it in an appropriate bitmap editor. If a color transition does not appear “smooth” enough, increase color resolution and save the file again.
Resolution, dots per inch	Field for image resolution. Since a bitmap file size is proportional to the resolution, a high resolution is only recommended when saving documents that contain fine details and high-curvature contours, as that will help render them with minimum losses. After saving the file, view it in an appropriate bitmap editor. If you are not satisfied with the quality of an image, increase the resolution and save the file again.

Tab. 11.1.4. Saving setup dialog for bitmap and PDF

Element	Description
Gray scale	This option serves to represent the color scale by the gray scale. The higher the Color Mode is set, the greater the number of grey tones will be.
Dimension images*	Page Image Extents in pixels. This value depends on the specified scale and resolution.
Embed fonts**	The option that allows getting the text to look the same in the PDF file as in the original document.

* The option is available in the dialog only when saving to bitmap.

** The option is available only in PDF saving dialog.

*** The option state does not affect the font appearance.

Tab. 11.1.5. Usage variants

Variant	Description
Black	All lines and text are saved in black; color fill and bitmap images are saved in grey scale.
Set for Item	Lines are saved in the colors which are set for them in System Settings dialog. Color fill and hatches are saved in colors defined when creating. 3D-object images are saved in colors defined when customizing properties of these objects.
Set for View*	Objects located in different views are saved in colors set for these views.
Set for Layer	Objects located in different layers are saved in colors set for these layers.

* Variant is available when saving drawings.

Parameter setup dialog for saving a model to PDF is presented in Fig. 11.1.2. The Table 11.1.6 presents the dialog controls.

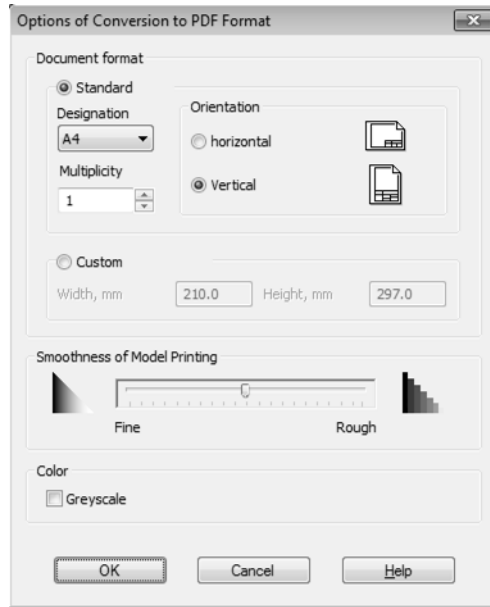


Fig. 11.1.2. Parameter setup dialog for saving a model to PDF

Tab. 11.1.6. Parameter setup dialog for saving a model to PDF

Element	Description
Document format	A group of options allows setting document sheet size. Detailed description of the controls is given in Table 9.1.61 on p. 1744.
Smoothness of Model Printing	The control allows changing model image output accuracy. To change the accuracy, move the slider across the range of positions from Rough to Fine .
Gray scale	The option allows you to record the shades of color by the shades of gray.

11.1.3.2. Multipage Drawing

Parameter setup dialog for saving a multipage drawing as bitmap or PDF is shown in the Fig. 11.1.3.

The main difference of this dialogs from drawing save dialogs (Fig. 11.1.1) is the **Range** group. Control elements of this group are shown in the Table 11.1.7. Other elements are described in Table 11.1.4.

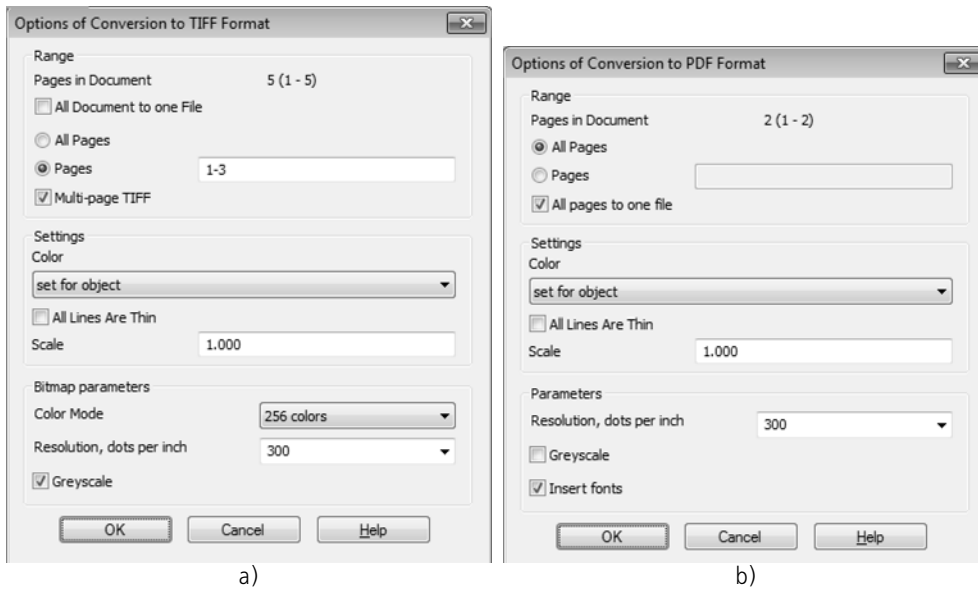


Fig. 11.1.3. Parameter setup dialog for saving multipage drawing:
a) bitmap, b) PDF

Tab. 11.1.7. Control elements of the **Range** group

Element	Description
Pages in Document	Total number of pages in a document.
All Document to One File*	This option allows placing all document pages on one page in TIFF file. For other formats, it allows saving all pages of the drawing in one file. If the option is disabled, separate bitmap file is created for each page when saving in any format except for TIFF. Additional symbols forming file names for separate pages, will be automatically added to the file name set for document saving. The method of saving in TIFF format with the option All Document to One File disabled is determined by the state of Multipage TIFF option.
All Pages	This option serves to save all document sheets (pages).
Pages	This option serves to specify page numbers to be saved. With this variant enabled, the field for page number entering becomes available.

Tab. 11.1.7. Control elements of the **Range** group

Element	Description
Multipage TIFF*	This option serves to create a single multipage file with one document sheet per page when saving a document in the TIFF format. If the option is disabled, a separate TIFF-format file is created for each sheet. This option is unavailable for any other format.
All pages to one file**	Option allows all drawing sheets be placed in one file. When the option is off, as many files as there are pages will be created.

* The option is available in the dialog only when saving to bitmap.

** The option is available only in PDF saving dialog.

11.1.3.3. Text Document, BOM

Parameter setup dialogs for saving text or BOM documents to bitmap or PDF is shown in the Fig. 11.1.4.

The main difference of this dialogs from drawing save dialogs (Fig. 11.1.1) — is the **Range** group. Control elements of this group are shown in the Table 11.1.8. Other elements are described in Table 11.1.4.

Note that when saving to any format other than TIFF, a separate bitmap file is created for each page in the document. The method of saving in the TIFF format is determined by the state of the **Multi-page TIFF** option in the **Range** group.

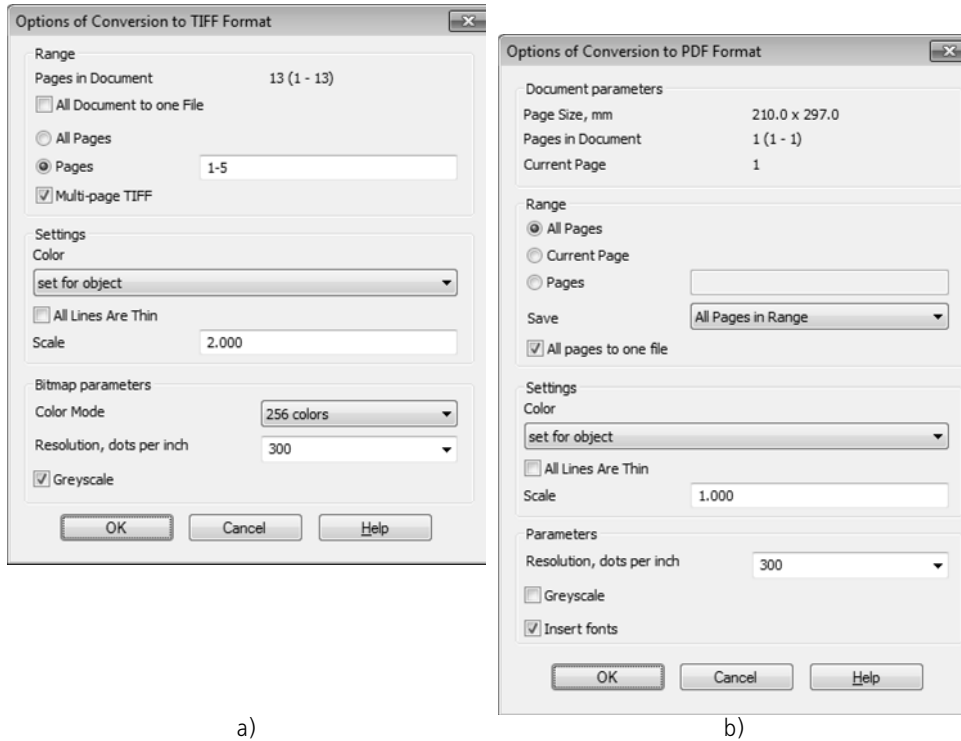


Fig. 11.1.4. Setup dialog for saving text and BOM documents:
a) bitmap, b) PDF

Tab. 11.1.8. Controls in the **Range** group

Element	Description
All Pages	Enabling this option saves all pages of a document.
Current Page	Enabling this option makes saving only the current document page— the page in which cursor was located before calling the saving command.
Pages	In this field, you can enter the numbers and/or range of page numbers to save. For example, “1, 3-5”.
Save	The list allowing you to specify what pages in the range need to be recorded: all , only the even or only the odd ones .

Tab. 11.1.8. Controls in the **Range** group

Element	Description
Multi-page TIFF*	This option serves to create a single multipage file with one document sheet per page when saving a document in the TIFF format. If the option is disabled, a separate TIFF-format file is created for each sheet. This option is unavailable for any other format.
All pages to one file**	Option allows all drawing sheets be placed in one file. When the option is off, as many files as there are pages will be created.

* The option is available in the dialog only when saving to bitmap.

** The option is available only in PDF saving dialog.

11.1.3.4. Conditions for saving to the bitmap format

A KOMPAS-document is saved into a bitmap format in two stages.

1. A temporary bitmap file with the 24-bit color depth is created. Maximum possible number of points in it is 1,431,655,724. Location of the temporary file is the folder defined by the *TEMP* variable of the KOMPAS-3D environment (or by the respective key of the *Kompas.ini* file).
2. The temporary file is converted into a file of the specified format with the specified resolution and color range.

Since maximum bitmap size is limited, the system compares the total number of points in the file being created with maximum allowed number of points upon clicking the **OK** button in the dialog of saving bitmap file parameters.

If the number of points is greater than the allowed number, a message is displayed on the screen: "Bitmap size exceeds maximum allowable size".

If, on the other hand, the number of points does not exceed the allowed limit, the system calculates the size of the temporary file. This value (in bits) results from multiplying the total number of points by 24, since, to represent the color in each point of the image, 24 bits are used (accordingly, multiplying by 3 gives the file size in bytes). The resulting value is compared with the disk space available for saving KOMPAS-3D system temporary files.

If there is not enough space for the temporary file, a message "Not enough disk space to create temporary file" will be displayed on the screen.

Therefore, if you received a message about exceeding the allowed bitmap size, the number of points in the image should be reduced. This can be done by reducing the scale or the resolution in the dialog of bitmap saving settings.

If, on the other hand, a message about the insufficient disk space is received, either the number of points in the image has to be reduced, or disk space available for saving temporary files must be increased.



You can get the information about storing temporary files in the Browse System Files and Folders dialog. To access the dialog, call the following command: **Service – Parameters... – System – Files – Location.**

11.2. Hyperlinks

11.2.1. General Information on Hyperlinks

It is possible to create hyperlinks in KOMPAS-3D graphic documents.

Hyperlink is the link between objects that allows to go from one object to another.

Hyperlink cannot exist in the document «as is» (i.e., separately from objects), that is why it is inserted in current object and becomes one of its properties. To create a hyperlink means to indicate an object that will contain it and an object to which you will be transferred (**target object**).

Transfer to the target object is done after **hyperlink activation**. By default, to activate a hyperlink, you must click the object that contains it while holding down <Alt>. The activation method can be changed when setting hyperlinks (see Section 9.1.2.7 on p. 1675).

Hyperlinks can be added to the following objects:

- ▼ geometrical objects,
- ▼ dimensions,
- ▼ designations,
- ▼ fragments and views insertions,
- ▼ macro elements, including library macros,
- ▼ tables and texts.

Target objects can be the following:

- ▼ file, folder, or web page,
- ▼ location in the current document,
- ▼ e-mail address.

One object may contain only one hyperlink. One document may contain several object with hyperlinks.

Hyperlink to the location in the document is **internal**. Going by internal hyperlink is performed to the area of the same document, for example, to the area of a view, a text, or an enlarged detail. Other hyperlinks are **external**. When following external hyperlink, a file, a folder, a web page, or an e-mail, Send window will open.

11.2.1.1. Hyperlinks Displaying and Activating

External view of an object is not changed after hyperlink creation. When you place the cursor over the object containing a hyperlink, a pop-up prompt with the information specific of each link type shall appear on the screen: path to the file, or web address, or the name of the item in the document, or e-mail address.



To activate a hyperlink, click the object while holding the <Alt> key. Mouse pointer is displayed as link pointer (Fig. 11.2.1). After clicking the hyperlink, the target object will appear on the screen: an open file, folder or web page, a portion of the current document, or an e-mail creation window.

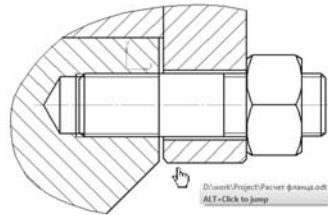


Fig. 11.2.1. Displaying Hyperlink

Information about hyperlink is displayed in the object properties table (see Section 3.4.1.2.1 on p. 1118).

The list of all external hyperlinks of the current document is included in the **Document summary** dialog on the **External links**. To call this dialog, the **Document information** command is used.

11.2.2. Working with Hyperlinks

11.2.2.1. Hyperlinks Creation



To create a hyperlink, select one or more objects and call the **Hyperlink...** command from the **Insert** menu or from the context menu.



Hyperlink Creation dialog also can be called by clicking **Hyperlink** string in the object properties table (see Section 3.4.1.2.1 on p. 1118).

The dialog will be displayed on the screen (Fig. 11.2.2), in which you should choose the hyperlink type and set its parameters. Buttons for choosing hyperlink type — **File**, **web page**, **Place in document** and **E-mail** — are located in the left part of the dialog.

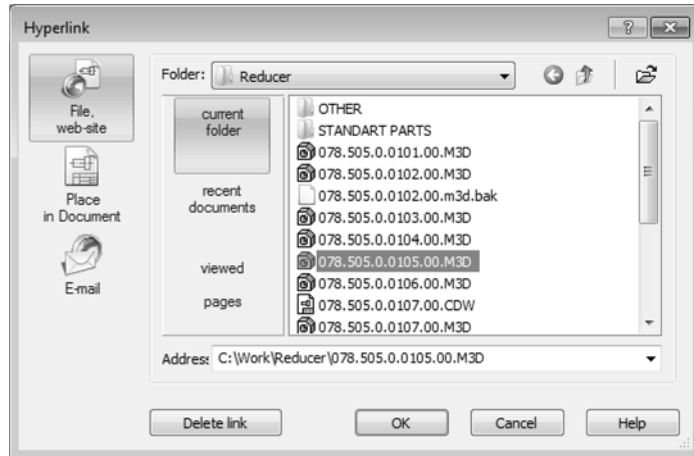


Fig. 11.2.2. Hyperlink Creation Dialog

After clicking the button, central part of the dialog changes that allows to set hyperlink parameters of any type. The description of hyperlinks creation is given in Sections 11.2.2.1.1, 11.2.2.1.2, 11.2.2.1.3.

If several objects are selected, each of them will contain the created hyperlink.

The **Hyperlink...** command is not accessible if no object is selected. The command is also inaccessible if objects with and without hyperlinks are selected at the same time.

Hyperlinks can be created automatically when setting some designations (see Section 3.3.14.2 on p. 1111).

11.2.2.1.1. Hyperlink to File or Web Page

Creation of hyperlink to file or web page is done while the **File, web page** button in the left part of the dialog is pushed (Fig. 11.2.3).

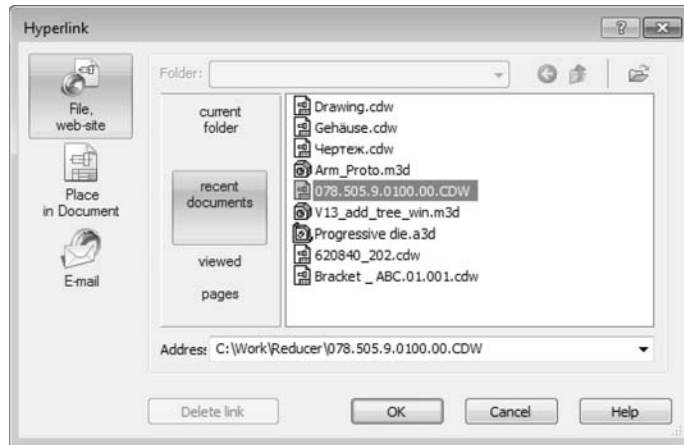


Fig. 11.2.3. Dialog of Hyperlink to File or Web Page Creation

If you need to create a hyperlink to a file or a folder, push the **Current Folder** button. Select the target object — a file or a folder — in the folder content view window. It is possible to change the current folder using the **Folder** list and **Back** and **Up** buttons.



To select the target object from the standard File Open dialog, click the **Open** button in the upper part of the **Hyperlink** dialog, and open the required file.

If you need to create a hyperlink to one of the last files, click the **Last Files** button and select the required file.

If you need to create a hyperlink to one of the last viewed web pages, push the **Viewed pages** button and select the required web page.

After selecting the target object, a path to it or its address appears in the **Address** field. You may type the path (address) on the keyboard or select the path (address) from the **Address** drop-down list used in the current KOMPAS-3D session.

Click **OK**. The button is available if the **Address** field is filled.

After creating a hyperlink, the pop-up tip contains the path to the file (folder) or the web address.

When activating a hyperlink, folders are opened by Windows Explorer, files are opened by the application which is assigned by Windows to this file type, and web pages are opened by a web browser.

11.2.2.1.2. Hyperlink to Place in Document

Creation of a hyperlink to a place in the current document is done with the pushed button **Place in document** in the left part of the dialog (Fig. 11.2.4).

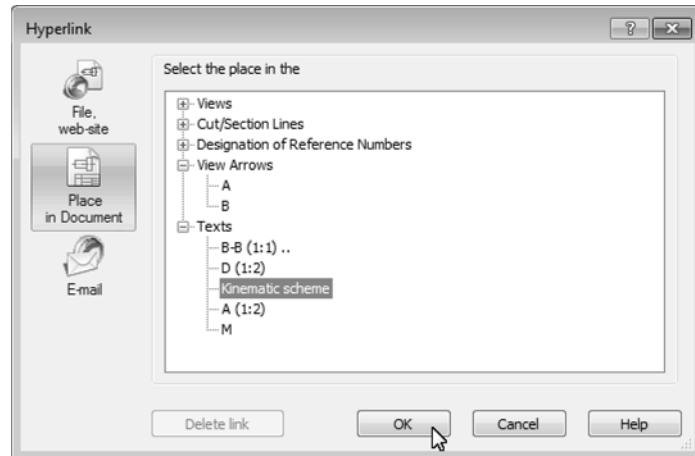


Fig. 11.2.4. Dialog of Creation Hyperlink to Place in Document

In the dialog window, groups of the current document objects to which it is possible to make a hyperlink are displayed. You can expand (collapse) groups by clicking the plus (minus) icon to the left of the group name. By default, groups are displayed in collapsed state.

Select the required target object from the list; the selected object is highlighted in the document.

Click **OK**. The button is available when the target object is selected.

After hyperlink creation, the pop-up tip contains the names of the document and the target object.

When activating the hyperlink, you go to the part of the document where the target object is located. The center of the target object is placed in the center of the screen.



If the target object of a hyperlink is in a hidden view or layer, it is impossible to activate the hyperlink.

11.2.2.1.3. Hyperlink to E-mail Address

Creation of hyperlink to an e-mail address is done when the **E-mail** button in the left part of the dialog is pushed (Fig. 11.2.5).

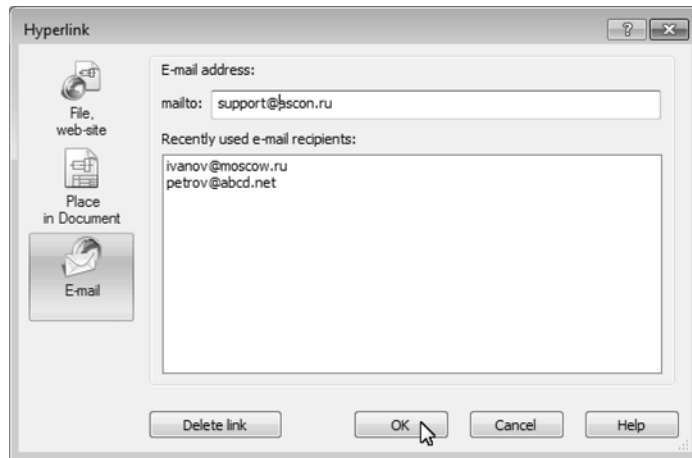


Fig. 11.2.5. Dialog of Creation Hyperlink to E-mail Window

Enter the address in the **mailto:** field or select it by double clicking from the list in the central window of the dialog. This list includes addresses used during the last working session of KOMPAS-3D.

Click **OK**. The button is available if the **mailto:** field contains the e-mail address.

After hyperlink creation, the pop-up tip contains the address of the e-mail message recipient.

When the hyperlink is activated, the program registered in the system as default e-mail application is opened. The window of e-mail message with automatically entered recipient address is displayed on the screen.

11.2.2.2. Editing Hyperlink

Editing hyperlink means changing its type and target object.

To edit a hyperlink, select the object that contains it and call the **Insert — Hyperlink...** command or the **Edit Hyperlink...** command from the context menu. You may edit several identical hyperlinks at a time after selecting objects that contain them.

The Edit Hyperlink command is unavailable if the selected object does not contain a hyperlink, or selected objects contain different hyperlinks.

After calling the command, the **Hyperlink** dialog is displayed in the state it was displayed during hyperlink creation. Enter new settings and click the **OK** button in the dialog. The dialog will close and the hyperlink will change.



Editing the target object does not affect the hyperlink if the results of editing are saved:

- ▼ for folder and file — the path and the name,
- ▼ for web page — the address,
- ▼ for an object of the current document — belonging to a view.

11.2.2.3. Deleting Hyperlink

To delete a hyperlink, select the object that contains it and call the **Delete Hyperlink** command from the context menu. You may delete hyperlinks from several objects at a time after selecting them.

The command is available if there is at least one object with a hyperlink among the selected objects.

You can also delete a hyperlink in the **Hyperlink** dialog when editing the hyperlink (see Section 11.2.2.2) by clicking the **Delete Hyperlink** button.



After deleting the target object, you cannot go to it using a hyperlink. After activating such a hyperlink, you get the message that the target object was not found.

11.3. Team work. Document recovery

11.3.1. Team work

11.3.1.1. Team work with KOMPAS–documents

If a KOMPAS document is opened on another workstation and you try to open it on your workstation, the message shown on Fig. 11.3.1 will appear on the screen.

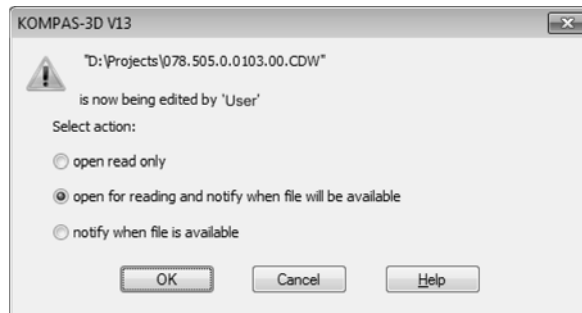


Fig. 11.3.1. A message that the document is currently in use



This and other messages mentioned in this section will appear if file change control is enabled in the access rights dialog (see Section 9.1.4.2 on p. 1694).

Otherwise, a document is opening in read-only mode. No messages appear when the document is being opened or when working with it.

You can select one of the operation versions:

▼ Open as Read-Only

Document is opening in read-only mode. The word “read-only” is displayed in the title of the read-only document window. Such document can be edited, but can not be saved in the same folder with the same name.

When the user who is working with the document changes it, you will receive messages (see Section 11.3.1.3 on p. 2027).

▼ open as Read-Only and Notify when the File Becomes Accessible

Document is opening in read-only mode. When the user, working with the document, will close it, you will receive the respective message that the document has become available for editing.

▼ If the document has not been edited, the message will look like on Fig.11.3.2, a. If you click **Yes**, the document opened in read-only mode will be closed and reopened in the mode available for editing. Clicking the **No** button will result that the document remains read-only.

▼ If the document has been edited, the message will look like on Fig.11.3.2, b. If the **save changes...** option is selected, you will be prompted to save the current document with

a different name, and then the document will be opened in the mode available for editing. If **open file...** option is selected, the current document will close without saving. Instead, the document, which has become available, will be opened.

▼ Notify when the file becomes accessible

The document cannot be opened but when it becomes available, you will receive a message having a look shown on Fig. 11.3.2, a.

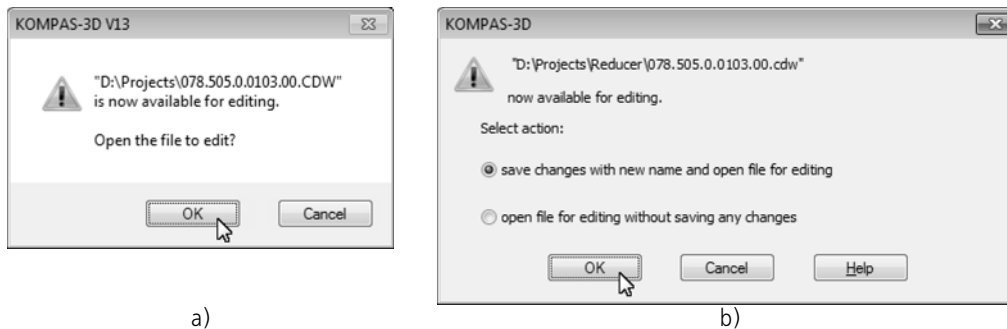


Fig. 11.3.2. Message on file accessibility: a) in the case, when read-only document was not edited, b) in the case, when read-only document was edited

If the option **Notify when read-only files are opened** is enabled in the access rights dialog (see Section 9.1.4.2 on p. 1694), the same message as shown on Fig.11.3.1 will appear also in the following cases:

- ▼ The opened file has a “read-only” attribute,
- ▼ The user trying to open the file has no permission to write or to add data in this file.

If the notification option is disabled, the document will be opened in the above-mentioned cases without notification.

11.3.1.2. Team work with linked files

Subordinate to documents are files of design libraries, attribute types, lines styles, hatches and texts, specification styles.

Using these files, i.e. applying designs, styles, attribute types stored in them, is possible from several desktops at the same time. Moreover, this approach ensures unification of documents created at different desktops.

The availability of **editing** subordinate files currently used at one of the workstations at a different workstation depends on the KOMPAS-3D settings for the workstation. This setting is made in the file access dialog (see Section 9.1.4.2 on p. 1694) using the **Level of access to subordinate files** list. It contains two options:

▼ Enable Reading.

This option means that the engaged subordinate files will be available at other workstations for use only. In addition, other user will be able to copy linked files.

▼ Read and Save Enable.

This option means that the engaged subordinate files will be available at other workstations

both for use and for editing. creating, editing and deleting layouts, attribute types, styles. When someone changes a subordinate file that you are using, you will receive notifications (see Section 11.3.1.3 on p. 2027).

11.3.1.3. Document and file change control

Enabling file change control and its regularity setting is made in the access rights dialog (see Section 9.1.4.2 on p. 1694).

If control enabled, the following cases of file change will be processed:

1. Read-only document edit

The message will be given having a look shown on Fig.11.3.3, a. It contains path to the file and two buttons:

- ▼ **Yes** — the document will be reopened. Remaining available for reading only it will display at your workstation in the format it was last saved by another user. All changes, if you made them in the document prior to re-reading, will be lost.
- ▼ **No** — will result in no changes in the opened document.

Thus, if you introduced any changes in a read-only document and these changes must be saved, click the **No** button and then save the document under a new name. If you did not edit the document or do not need to save any changes, click the **Yes** button.

2. Change insertion source - a fragment, view or component

The message will be given having a look shown on Fig.11.3.3, a. It contains path to the source file and two buttons:

- ▼ **Yes**— the insertion source will be immediately re-read. For example, another user has edited a fragment inserted in the document you are currently working with. The re-reading of the source will result in the redrawing of the insert image.
- ▼ **No**— no immediate re-reading. When the document containing an insert is opened next time, all the changes made in the insertion source will be unconditionally transferred to the document being opened.

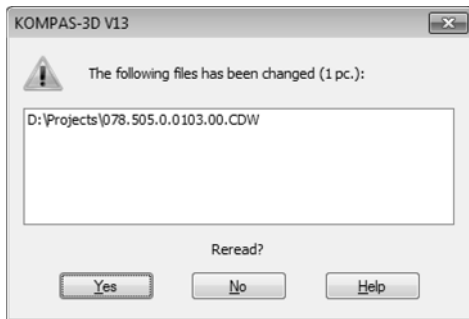
3. Linked File Edit

The message will be given having a look shown on Fig.11.3.3, b. It contains path to the file and two buttons:

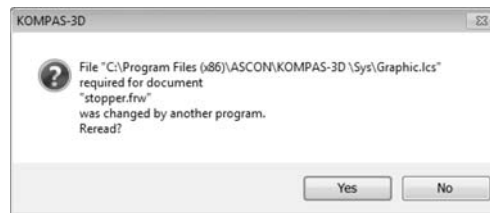
- ▼ **Yes**— changed style, attribute type or design will be promptly received from the subordinate file. For example, another user has edited the line style library by changing the color of Style No. 5. As a result of re-reading the library in the document, that is opened on your PC, the color of lines of this style will be changed.
- ▼ **No**— no re-reading of subordinate file. This only means that re-reading will not be performed immediately. When the document that uses the subordinate file will be opened next time, all changes, made in this file, will be displayed in the document to be opened unconditionally.



Re-reading design libraries (*.lyt files) is made in accordance with the design update setting (see Section 10.2.1.2.1 on p. 1961) which was applied when the document was being opened.



a)



b)

Fig. 11.3.3. Message: a) of changes of the model that is opened for reading, b) of changes in a line style library

11.3.1.4. Satellite files

Once a KOMPAS document has been opened for editing, its satellite file is automatically created.

The satellite file is a service file located in the same folder as the document to be opened.

The satellite file name is formed in accordance with the following rules:

- ▼ ~\$ symbols are added to the document name and come first, if the document name contains not more than 6 symbols;
- ▼ If the name contains more than 6 symbols, ~\$ signs replace the first and the second symbols;
- ▼ ~ sign replaces the last symbol of extension of document file.

For example, *SAM.CDW* document will have a satellite file *~\$SAM.CD~*, while *SAMPLE_1.CDW* document will have a satellite file *~\$MPLE_1.CD~*.

The presence of a satellite file in the folder means that the KOMPAS document is already being used and, once opened, will be available for reading only.

The satellite file contains textual information about the user who has opened the document for editing, as well as the number which identifies the document at the moment of its opening.

The satellite file will be deleted from the files folder automatically after the document, which has been opened for editing, is closed.

In case of an emergency exit from the system, satellite files will not be automatically deleted. Sometimes they can prevent reopening of the document (read-only file opening message appears). In this case, it is recommended to delete satellite files manually using the respective Windows utility.

When the KOMPAS document is opened for reading, no satellite file is created.

11.3.2. Backup copy

11.3.2.1. Backup copy files

Backup copy file is created when saving an edited KOMPAS document and contains previous version of this document, i.e. the document as it was prior to the editing. If a document is being saved for the first time, a backup copy file is not created.

When setting up KOMPAS-3D (see Section 9.1.4.3 on p. 1695), saving earlier copies of documents can be enabled or disabled. If saving of earlier copies is enabled, they can be located in the same folder with the documents or in a different folder.

Saving the Document occurs as follows: After calling the **File — Save** command, previous document version is recorded into a backup copy file, and then new version is recorded into a document file.

- ▼ If previous copies saving is disabled, then after finishing saving the document the backup copy file is deleted. In case of an emergency closing of KOMPAS-3D when saving a document, the backup copy file will not be deleted. It is saved in the same folder with the document. The name of the backup file is formed in accordance with the following pattern: <document's file name> + <.> + <name extension> + <.> + <bak> (for example, *example.frw.bak*).
- ▼ If previous copies saving is enabled, then
 - ▼ when saving the copy in the same folder with the document, the backup file name is formed in accordance with the pattern above.
 - ▼ when saving the copy in a different file, the name and extension of the backup file will be the same as those of the document file.

Therefore, in case of document loss or failure of KOMPAS-3D when saving a document, the information can be **partially** restored from the backup copy files. Backup copy file with the *bak* extension opening is detailed in Section 11.3.2.2.

11.3.2.2. Document recovery from the backup copy files

To open the backup file, remove the *bak* extension of the file using Windows Explorer, start KOMPAS-3D and open the file in a normal way using **File - Open**.

KOMPAS-3D is not associated with *bak* extension files as a program used to open them by default. Therefore, double-clicking on the **.bak* file in the Windows Explorer does not start KOMPAS-3D automatically. A dialog will be displayed to select the program. Specify the executable file of KOMPAS-3D (*KOMPAS.exe*) as a program for file opening; then the system will be started and the file will be opened in it.



It is not recommended to enable the **Use for all files of this type** option in the program selection dialog window. Backup document copy files with *bak* extension are generated by different programs. If KOMPAS-3D is associated with *bak* files for opening then an error message will be displayed when you attempt to open a *.bak* file created by another application.

The second way to open the backup file is as follows: Start KOMPAS-3D and call **File — Open**. In the **File Type** list of the file opening dialog appeared choose the **All Files (*.*)** row, in the

file list— desired file with *bak* extension. KOMPAS-3D will determine the type of the document that is saved in the backup copy and open it. Save the opened file with the **File — Save As...** command. An extension matching the document type will be assigned automatically to the saveable files.

11.3.3. Auto Save

11.3.3.1. Auto Save Files

Auto save files are created as follows: If auto save is enabled (see Section 9.1.4.4 on p. 1698), the auto save file will be created upon expiration of the preset period for the document which is being edited in the current session. It contains the document with modifications made to it after it was created and after latest calling the **File — Save** command, i.e. when it was saved to disk for the last time.

Names of auto save files are formed automatically using the following array:

<k> + <version number of KOMPAS> + <a random set of six Latin characters>.

Extension of the auto save file name is formed in accordance with document type. The final symbol of the extension will be replaced by \$ symbol.

For example, autosave file for a fragment to be edited in KOMPAS version 9 system can have the name *k9cxfhme.fr\$*.

Upon expiration of another preset time interval, auto save file content will be compared to the current document. If changes are found, the auto save file will be re-written. If there is no modifications, file remains unchanged.

When saving a document using **File — Save** command, the auto save file will be deleted and the cycle will be repeated.

In case of an emergency shut-down of the system, auto save file is not deleted. When the KOMPAS-3D is started again this file is automatically used for the document recovery.

11.3.3.2. Restoring documents from Auto Saved files

Restoring documents after emergency shut-down of the system is performed automatically using auto save files.

To do this, start KOMPAS-3D by any standard method except for double-clicking a file in Windows Explorer that was opened or being edited at the moment of the application failure.

Why shouldn't KOMPAS-3D be run by double-clicking the file name? Because in this case the document will not be restored. The file will be opened, in which the document is saved as it was at the moment of the latest calling the **File — Save** command before the emergency shut-down (at the moment of the latest saving to disk). Obviously, if a comparatively short auto save period had been set up, the auto save file will most probably contain more changes than the last version of the document saved manually.

After the system is loaded, auto save files are correctly searched for in the folder specified for such files saving. Using all files found, the respective documents are restored.

“Restored” status appears in the heading of the restored document window. This status is displayed until the first editing of the document.

To save the restored document to disk, call **File — Save** command (to save to the same file which was used before the emergency shut-down) or **File — Save as...** (for saving to new file), the autosave file from which the document was restored is deleted.

If the system has been run by double-clicking the file which was being edited at the moment of an emergency shut-down (let us call it “file X”), the document (document X) saved to that file will not be restored; document X will be opened as it was saved to disk the last time. If other documents were opened in KOMPAS-3D as of the moment of the application failure, they will be restored. All autosave files are deleted except for the one that matches the document. This autosave file remains in its original place and is not changed no matter what changes are made to document X.

Thus, the following actions must be performed after the “wrong” running of the system.

1. Save all the recovered documents.
2. Close document X without saving.
3. Close the system.
4. Run the system using alternative method.
5. Save document X restored from the auto save file which relates to file X.

12. Solutions to applied problems

12.1. Application libraries and document libraries

12.1.1. Library Manager

Use Library Manager to control application libraries and document libraries KOMPAS-3D.



To enable or disable the Library Manager Window, use the **Service — Library Manager** command.

12.1.1.1. Library Manager Window

Library Manager window may have several tabs. On the first tab - **KOMPAS Libraries** the structure of manager is shown: lists of sections and libraries (Fig.12.1.1).

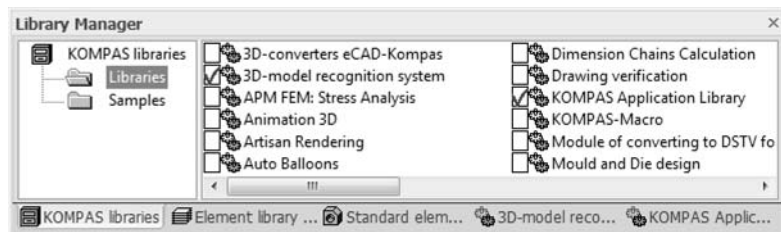


Fig. 12.1.1. Library Manager Window

If a section has connected libraries, then its icon is displayed gray, if not — blue.

To the left from each library name an icon designating this library type is located:



- ▼ Application Library
- ▼ Fragment Library
- ▼ Model Library

Other Manager tabs display the contents of the currently connected libraries of fragments, models and application libraries working in the panel mode.

Attached libraries are marked with a red checkmark.

To work with the Manager and libraries, you have commands of the tabs context menus.

12.1.1.2. Library Manager control

Commands of sections view range context menu are designed for controlling of Library Manager (Fig.12.1.1). These commands are shown in the table 12.1.1.

Tab. 12.1.1. Library Manager control commands

Command	Description
Add	Adding library to the Library Manager.
Description	

Tab. 12.1.1. Library Manager control commands

Command	Description
View	Setting the appearance of the Library Manager panel.
Create Section	Creation of a new section (subsection) in the Library Manager.
Delete	Deletion of the selected section in the Library Manager. A section in the Manager is deleted with all its subsections.
Insert	Paste library from the clipboard into selected section of the Library Manager*. If the clipboard is empty, the command is not accessible.
Update Library Manager	Deletion of libraries containing files that are not found at defined paths, and adding libraries indicated in files *.lms that are located in the subfolder Sys of the system main folder**.
Clear Library Manager	Delete all libraries except those that are attached at the moment of the command calling from the Manager.

* The library manager uses its own proprietary mechanism of the clipboard, which is different from the standard Windows clipboard.

** The command is also available in the section **Service** of the KOMPAS-3D main menu.

12.1.1.3. Library Management

Context menu commands of the library selected in the list are designed for library management (Fig.12.1.1). These commands are shown in the table 12.1.2.

Tab. 12.1.2. Library management commands

Command	Description
Run	Start the library selected in the list. The libraries of fragments and models are also launched when connected (linked).
Attach	Attach the library selected in the list. Attaching is needed for library features becoming available to the user. The command is available for non-connected libraries. The selected library is connected in its specified working mode. A red “check” appears next to the library name — meaning that the library is connected. To detach the library, repeat command Attach . You can also click on the field with the red “check” next to the name of the library.
working mode	Select the operation mode for the library selected in the list.

Tab. 12.1.2. Library management commands

Command	Description
Properties	Set properties of the library selected in the list. After calling the command, the Settings dialog will appear. The command is available for non-connected libraries.
Delete	Delete the library selected in the list from the Library Manager*.
Cut	Delete the library selected in the list from the Library Manager and place it to the Manager clipboard. Cutting of a subsequent library and placing it to the Clipboard means deleting its previous content.
Insert	Paste a library from the clipboard into the current Manager section.
Disable all	Detach all attached libraries from the KOMPAS-3D system.

* Deleting the library means detaching access to it from the Library Manager. In this case, no operations with the library file on disk are performed.

12.1.2. Fragment Library

When working with KOMPAS-3D, you can save created images of standardized parts in fragments and then paste them into new drawings. For repeated insertion of the same fragments into the drawing, it is more convenient to use **fragment libraries**. It is possible to organize and store different standard fragments with optional comments to them in the libraries. Using fragment libraries facilitates finding ready images and inserting them into the document.

Each fragment library is a separate file with the *lfr* extension. The library fragments are not separate files on disk, rather they are part of a single library file. Fragments are stored in the way of ordered lists in subsections and in the root section of the library. The names of fragments and library models can use any characters, the number of fragments and sections is not limited.

The KOMPAS-3D package includes some fragment libraries (i.e., technologic symbol library). To work with fragment libraries that are not included into the package, a separately paid license is needed.

Each fragment library is displayed on a separate tab of the Library Manager (Fig. 12.1.2).

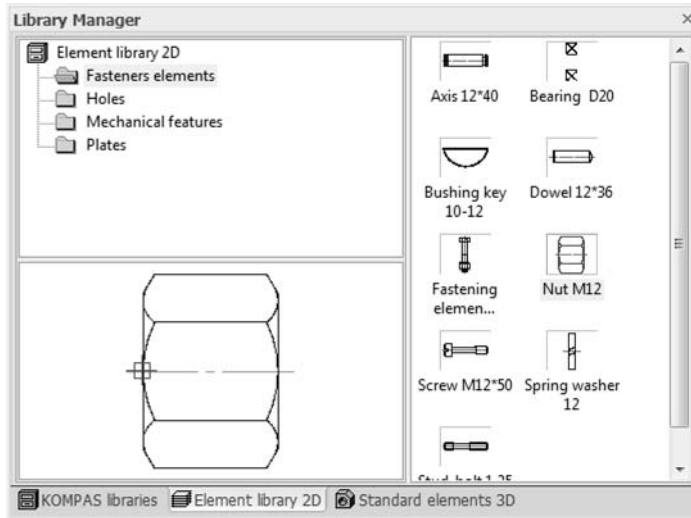


Fig. 12.1.2. Fragment Library Example

To set up tab appearance, the command **View** from its context menu is used.

12.1.2.1. Creation of Fragment Library

To create a new fragment library, do the following.

1. Call the command **Add Description – Document Libraries** from the Manager list of sections context menu.
2. In the displayed dialog, enter the name of new library and confirm its creation.
3. In the displayed dialog of library properties, enter its name in the Manager.
The created library will appear in the Manager list of libraries. This library is not attached yet.
4. Select the created library and call the command **Attach** from the context menu.
A tab that corresponds to the library you have created will appear in the Library Manager window. This tab is empty because newly created library does not contain fragments yet.
5. Form the library structure using the command **Create Section** from the tab context menu.
6. Include fragments into sections. You can do this by the following methods:
 - ▼ place ready fragments to the library,
 - ▼ create fragments directly in the library.

If there are ready fragments that will be saved in the library, do the following.

- 6.1. Call the **Add Fragment to Library...** command from the tab context menu.
- 6.2. In the displayed dialog, indicate directory and file name (s) *.frw to be included to the library.
- 6.3. In the displayed dialog of the fragment name setting, set the default name (full or relative) or enter optional fragment (s) name in the library.

Selected fragments will be placed to the current library section with set names.

If there are no ready fragments, do the following.

- 6.4. Call the **New Fragment** command from the tab context menu.
- 6.5. In the displayed dialog, enter the fragment name in the library.
KOMPAS-3D will open a new fragment window.
- 6.6. Create image that will be saved in the fragment.
- 6.7. Save and close the document.

Created fragment will be placed to the current library section.

For any further library fragment editing, select it in the list and call the **Edit** command from the context menu.

To manage the library structure, use the **Delete**, **Copy**, **Cut** and **Paste** commands of the context menu. Using them, you can move fragments and subsections between sections, and delete sections and fragments that you do not need any more.

Besides, using copy, cut and paste commands allows to exchange sections and fragments between various fragment libraries.

12.1.2.2. Fragment Insertion from Library

To insert fragment from library to the current graphic document, do the following.

1. Select the fragment to be pasted in the list and call the **Insert Fragment to Document** command from the context menu. You can also double-click the name of the required fragment with the mouse.

The system will switch to the fragment insertion mode. The Property Bar will display controls of insertion management (see Table). 3.7.2 on p. 1239).



2. Set up paste settings at Property Bar and define the anchor point of the fragment.
3. Make required number of insertions and click **Interrupt Command** button.

12.1.2.3. Detaching Library

If any fragment library is not needed any more, call the **Close** command from the context menu of the tab with the same name. The library will be detached from the KOMPAS-3D system.

12.1.2.4. Utilities

When working with a fragment library, utilities are accessible through the tab context menu commands. These commands are shown in the table12.1.3.

Tab. 12.1.3. Library Utilities

Command	Description
Optimize	<p>Library Optimization.</p> <p>Deleting fragments from the library does not change its file size on the disk. The place in the file where removed fragment was saved remains free. While creating, the next fragment in this library is saved into a free space in the library file. If there is no empty space, the library size is expanded.</p> <p>Sometimes, a library file that has a small number of fragments may have a large size.</p> <p>The Optimize command serves to adjust the file size to the number of models (fragments) stored in it. After calling the command, all free portions are excluded from the library file*.</p> <p>When a library file size is significant, the execution of the command can take several minutes.</p>
Include External Links	<p>Copying to the library all objects that exist in it as links to files:</p> <ul style="list-style-type: none"> ▼ Attribute types from attribute type libraries ▼ Styles of lines, hatches, texts from corresponding libraries ▼ Fragments inserted using external reference. <p>The list of references can be viewed in the document ** info dialog** on the External References tab while editing library document.</p> <p>No relation is maintained with libraries and fragment source files after executing the Include External References command. Included references are removed from the list of external references in the document summary dialog.</p> <p>References to template libraries (they are presented in the document which has BOM items) as well as the model's references to its components cannot be included. These links remain in documents, i.e. after call of Include external links command they are still displayed in the document information dialog. The paths displayed in the document summary info should point to existing files on which library documents refer to.</p>
Save fragment to File As...	<p>Saving of the library fragment in a separate file on a disk.</p> <p>Upon calling the command, a dialog is displayed, in which you need to specify the folder and the file name for saving.</p>
Info	Displays the dialog of the current fragment information.

* Optimizing library models may be compared to defragmentation of a hard-drive conducted for a single file.

** Click **File— Document Information...** to open the info dialog.



The last two commands are available only in the library fragment context menu.

12.1.3. Application Libraries

Application library is an application created to extend standard KOMPAS-3D capabilities and running in its environment.

Application libraries provide a programming language access to all KOMPAS-3D capabilities available in the interactive mode (creating and editing objects, work with a model document, opening and saving drawings and fragments, etc.).

Application libraries are included into the KOMPAS-3D package. To start operating the library, you must enable it in the Library Manager window (see section 12.1.1.3 on p. 2036). The library commands can be called using a standard method (from a menu, dialog, window, bar) or by clicking buttons of the toolbar or the custom toolbar (on custom toolbar creation, see Section 9.4.2.2 on p. 1874).

To operate the **KOMPAS Application Library**, **Document Check**, **FTDraw Library** and some other libraries, no separately paid license is required. They are located in the **Others** section on the **KOMPAS Libraries** tab of the Library Manager (see section 12.1.1.1 on p. 2035). To connect the selected library quickly, click on the field next to its name, whereupon a red “check” appears in this field— the signal that the library is connected.

12.1.3.1. Service tools

The **Service Tools** contains commands to perform various operations in graphics documents, sketches, and BOM. Commands that are similar in what they do are united in sections.

The commands are presented in Table 12.1.4.

Tab. 12.1.4. Description of Subsections of the KOMPAS Application Library

Section/command	Description
Operations with specification	<p>The commands of this section allow you to perform the following operations on a specification:</p> <ul style="list-style-type: none"> ▼ to convert to a fragment (*.frw files), DB tables (*.dbf files) or MS Excel (*.xls files), ▼ to process numeric columns— to assign variables to the values in columns and connect them with equations to perform arithmetic operations with them. <p>The conversion results in new file creation— a fragment or a table, in which data from a specification are inserted.</p> <p>As a result of processing numeric columns, the calculated values are entered in a specification.</p>

Tab. 12.1.4. Description of Subsections of the KOMPAS Application Library

Section/command	Description
Hole	The section contains one command, Hole , that allows an image of blind, through, smooth, and threaded holes of the following types to be created: <ul style="list-style-type: none"> ▼ Simple, ▼ Conical, ▼ With countersink, ▼ With counterbore, ▼ With countersink and counterbore.
Closure check	The commands in this section allow identifying open contours among the included in the document.
Contour approximation	The command allows approximate curved lines (contours, equidistants, Bezier curves, NURBS) by contours consisting of arcs and/or intervals.
Curve approximation	The command allows approximating contours by polylines and NURBS curves;
Selecting dimensions with manual entry	The command allows dimensions to be identified whose nominal value was input manually, not set automatically.
Profile MP calculation	The command allows calculating mass-center characteristics of the profile whose section is imaged in the document, and place a table with calculation results in the document.
Replace text	The command allows replacing one text fragment with another in unspecified surface roughness designations, terms of reference, tables, title block, and other texts in a graphics document, and specification objects.
Associate dimensions	The command allows form associative links between dimensions and objects for which they are specified.
Align objects	The command allows aligning geometric objects by extending them until they intersect.
Hide selected objects	The command allows turning off the display of selected objects transferring them onto an automatically created hidden layer.
Contour along area borders	The command allows plotting a closed contour for the intersecting curves that are the closest to a specified point.
Contour along two contours	The command allows plotting a closed contour by uniting, subtracting, or intersecting two specified contours.

12.1.3.2. Graphic document checking (Document checking)

The **Document Check** application allows you to find in KOMPAS-3D documents (drawings, fragments and sketches) possible errors:

- ▼ overlapping or superimposed segments, circles, arcs;
- ▼ designation of reference numbers having no links with specification objects;
- ▼ linear dimension lines crossing other lines of a drawing;
- ▼ distances between parallel dimension lines in case these distances are less than a preset value.

After calling the library command, you must set checking terms. After the command activation, detected errors can be automatically corrected.

12.1.3.3. Creating graphs (FTDraw Library)

The **FTDraw Library** application allows to perform the following actions in KOMPAS system documents:

- ▼ to plot functional dependence diagrams in Cartesian and polar coordinates with possible table presentation of these functions within the predefined range;
- ▼ to create a Bezier curve by the points which coordinates are tabulated as well as to present the Bezier curve available in a document as a point coordinate table;
- ▼ to perform elementary mathematical calculations.

To plot a diagram, enter a function (or select a table of Bezier curve coordinates— the **.Ftt* file), set the plotting parameters and specify the diagram location in a document once the command is called.

12.1.4. Sketches Library

One of the provisions to automate design is creating user's sketch libraries. For instance, you can form a library of parametric sketches that contain contours for key grooves, and then use them when creating shaft models.

In such a case, you would not have to create the same sketches many times. You can use a once created image saved in a library, possibly a parametric one, in any new sketch to develop.

The user's library of sketches is a common library of KOMPAS-3D fragments (a file with extension *lfr*).

It is created in the same way, as any other library of fragments (see Section 12.1.2.1 on p. 2038).

If fragments are created in the library with the purpose of used as sketches of 3D elements, those should satisfy the requirements to sketches for particular operations.

The fragments that contain sketches can be parametric.

12.1.4.1. Connecting Library of Sketches

To use the created library of fragments as a library of sketches, it is necessary to indicate the route to this library in the dialog, called with the **Service— Parameters... — System— Model**

Editor— Design Element Libraries command. The dialog is described in Section 9.1.11.4 on p. 1728.

12.1.4.2. The Use of the Library of Sketches

If you need to create a sketch on the selected plane or flat face, which is not a new one, rather, existing in the library, call the context menu command **Sketch from the Library**.

After that the Properties bar will display the library of sketches, the name of which was indicated in the system setup dialogue (Fig. 12.1.3). The sketch highlighted in the list is displayed in the view window in the Properties bar.

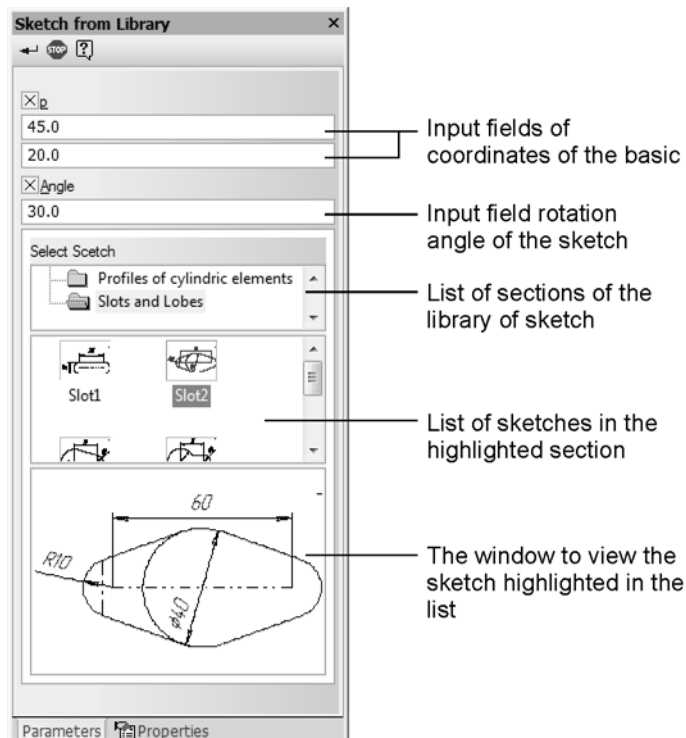


Fig. 12.1.3. Selection of a sketch in the Properties bar

The ghost of the sketch selected in the library is displayed in the Model window. Then, the coordinates origin of the library fragment is aligned with the origin of the plane or face where the sketch is located.

You can change the sketch position on the plane (translate or rotate it). The Property bar has fields for defining the position of the library sketch origin in the sketch plane coordinates (**t**) and the sketch rotation angle (**Angle**).

After setting the required position for the selected sketch, click button **Create Object**. Graphic objects from the library fragment will be copied into the new sketch, and the system will return to the part mode.

Icon of the created sketch will be displayed in the Model Tree.



A sketch inserted from a library loses references to its source. Its further editing process is the same as the process of editing other sketches.

12.1.4.3. User Library of Holes

If a round hole is created in a 3D model using command **Hole** (see Section 2.3.9.2 on p. 215), to form it, the sketch of the hole profile is used. This sketch is a fragment from the library (see Section 12.1.2 on p. 2037). By default, the full name of the library is `...Program Files\AS-COMKOMPAS-3D V...Sys\Holelib.lfr`.

You can create your own library of fragments, in which sketches for round holes shall be stored.

Sketches which are to be used when performing command **Hole from library**, should meet the following requirements:

- ▼ The axis of revolution must be a drawn by a line segment with a *Centerline* style.
- ▼ There should be a single axis of revolution.
- ▼ The rotation axis should pass vertically through the origin of the fragment.
- ▼ The insertion point of the hole (the point of intersection of its axis and the plane where it is based), should coincide with the origin of the fragment.
- ▼ The sketch can contain only one contour drawn using the line style *Main*.
- ▼ This contour should be located on one side of the rotation axis.
- ▼ The contour must be open.

To store sketches of hole profiles with different topology, different fragments should be used.

To realize insertion of holes with the same topology and different parameters, the fragment can be made parametric.



Sometimes, for parameterization of a fragment it should include the image of the hole profile on each side of the rotation axis. In this case, apply to geometrical objects on one side of the axis the line style *Thin*, and on another side— *Main*.

To use the created library of fragments when performing command **Hole**, it is necessary to change system settings and indicate the route to this library (see Section 9.1.11.4 on p. 1728).

12.1.5. Models Library

Often in different assemblies models are used, that differ only in values of their parameters— typical models (usually these are simple parts, like bushes, rings, etc.).

When the model from the disk in this assembly is inserted into the assembly, the link to its file is created, instead of copying the inserted model. Therefore, the model delivered to the assembly from the disk, can only have such parameter value, with which it is recorded in its file. Thus, for insertion of typical models that have different combinations of parameter values, it is necessary to have on the disk the number of files for these models equal to the number of inserts to be made.

However, it is quite unnecessary to create lots of files of typical models with various combinations of parameter values. Instead, you can build one parametric model and change its parameters when inserting it in various assemblies. Such models are recommended to be stored in libraries of models.

Different inserts of a model from the library can have different values for the same parameters. In this case, the source model in the library does not change.

Each model library is a separate file with the *.l3d* extension. The library models are not separate files on disk, rather they are part of a single library file. The list of models in the library can be structured.

You can add various models to libraries, regulate them based on a certain attribute and introduce random comments to them. Using libraries, you can perform multiple insertions of models into assembly documents. The use of libraries of models simplifies search of ready components and their insertion into the assembly, and considerably accelerates creation of assemblies that contain typical parts and subassemblies.

Methods to work in the library of models (creation of a library, sections and models in it) do not differ from methods to work with a library of fragments (see Section 12.1.2 on p. 2037). While working in the library of models, instead of commands **Add Fragment to Library...**, **New Fragment**, use commands **Add Model to Library...**, **New Detail/Assembly**, correspondingly.

12.1.5.1. Peculiarities of Library Models

To enable editing a model when inserting from a library and later on (modify its dimensions and topology), it should have external variables. (see Section 7.1.5.3 on p. 1593).

12.1.5.2. Adding Library Models to an Assembly

A model from a library can only be inserted in an assembly document. To insert a document into the current assembly, take these steps:

1. Highlight the model in the library window and call from the context menu the command **Insert model into assembly** or double-click the model in the library window.

A ghost of the model being inserted will appear in the current assembly window, and on the Properties panel—the **Location** and **Properties** tabs.

2. On the **Location** tab, perform the following actions.
 - ▼ Set the position parameters of the model in the assembly. The steps for a model from a library are the same as for a component (see Section 2.11.2.5.2 on p. 692).
 - ▼ If the model selected for insertion contains external variables, you can change their values. In this case, the Properties panel will have the **Variables** toolbar. It contains a list of all external variables of the model. The external parameter values are shown as they were at the time of last saving of the model in the library.

You can change the value of a variable by double clicking the respective **Value** field and typing in the desired value or using the counter. The variable whose value has been changed gets marked with a tick mark. Clicking the tick mark restores the initial value of the variable.



The values of variables can also be changed using a table of variables (see below).

3. On the **Properties** tab, set the color and optical properties of the component (control elements used for this are described in Section 2.14.2 on p. 784), turn on/off specification object creation.



If the model contains a variable table, you can also select new values of the external variables using the table. For this, you need to click the **Table of Variable** button and select the required row in the table that appears in the screen and click **OK**. External variables of the model will receive values recorded in cells of the selected line.

Button **Table of Variables** is available, if the table of variables stored in the inserted model, contains more than one line. The table of variables is built in the file during its creation or editing, and is saved in it. More detailed information on variable tables can be found in Section 7.1.5.4 on p. 1594.



An assembly can include several inserts of the same model with different values of respective variables (and, therefore, with different dimensions and topology). This is possible because external model variables and equations relating them with other variables are stored in the insert itself. Since each variable controls its part of the assembly, those do not conflict, not “confronting” each other.



4. In order to complete insertion of a library model, click the **Create object** button.



The Model Tree will display an icon of a user’s model library component.

If you need to insert the same model again, repeatedly call the command of inserting library models. You can also use the method of copying components using the mouse (see Section 2.14.4.4 on p. 800).

Sometimes, upon inserting, a library component is marked in the Model Tree as failing. In this case, the library model is displayed only in the Tree, while not being visible in the assembly in window.

A reason for a library component failure can be one of the following:

- ▼ Entering such values of external variables, which prevent the system of equations and inequalities relating model variables from solving.
- ▼ One or more model variables are assigned values outside their valid range.
- ▼ Error in a model of the library (i.e., the model inserted from the library contained an error even at the stage of creation of this library).

If the model contains external variables (i.e. if the error can be caused by the first or second reasons), edit value of variables for this model (see Section 2.14.3.3.1 on p. 789) or restore them.

If the error was not fixed by changing variable values (or if there are no external variables in the model), then that error was caused by another reason, and to correct it you need to edit the model in the library. To edit a model in a library, connect that library, select the failing model and call the context menu command **Edit**. After editing save the model, close its window and insert the model in the assembly again.

12.2. Using OLE Technology

12.2.1. General OLE Technology Data

Object Linking and Embedding (OLE) is Windows' one of the most powerful tools allowing joint running of and data exchange between various applications.

OLE technology helps user to insert objects, created in various applications, to a Windows document. For example, it is possible to embed a KOMPAS-3D drawing or fragment to the Microsoft Word text document. Having embedded an object (document), it can be edited using commands of “native” application (i.e., to edit a drawing or fragment embedded to Word, the KOMPAS-3D system will be activated).

You can insert only integral KOMPAS-files to other documents. Insertion of single objects of the KOMPAS-3D through the Clipboard is not supported (KOMPAS-3D uses its own clipboard, which is different from the standard Clipboard).

You can embed an item and at the same time maintain connection with the source file and allow auto update of the embedded object after any modifications of the source file.

Further on, application of the OLE mechanisms will be described based on interactions between KOMPAS-3D and Microsoft Word. Joint operation with other Windows applications is almost the same, however, you can find minor differences related with specific features and dialogs messages of these applications.



Display of an OLE insert is controlled by print filter settings, i.e. the document containing the insert shows only those source file objects which printing is enabled. See Section 9.1.5.2 on p. 1701 for the print filter settings.

12.2.2. Working with Insertions

12.2.2.1. Insertion of a KOMPAS-document

To insert a KOMPAS-document to Word, complete the following steps.

1. In Word: call command **Insert** — **Object...**
2. In the displayed dialog, activate tab **Create from File**. If you need to display an icon of the embedded KOMPAS-document in a Word document, enable option **As Icon**.
3. If you need the embedded KOMPAS-document to maintain connection with the source file, enable option **Connection with File**. Any further modification to the source file will be displayed in the document containing the inserted object.
4. Enter full name of the inserted file to field **File Name**. You can also click the **Browse** button and specify the required folder and document file.
5. Click **OK** in the **Insert Object** dialog.

The content of the selected file will be inserted to the Word document.



OLE-insert of the multi-sheet KOMPAS-document (a text document, BOM or multisheet drawing) contains only the first sheet of the document.



Pay a special attention when renaming or transferring files being sources for the connected inserts, since there may be a risk of breaking such connection.

12.2.2.2. Editing Embedded KOMPAS-document

To edit the embedded KOMPAS-document, complete the following steps.

1. Click the right mouse button on the inserted KOMPAS-document.
2. In the appeared menu select **<KOMPAS-document type> Object...**

A submenu containing commands **Edit**, **Open** and **Modify** will appear.

If the inserted KOMPAS-document is connected with the source file, activation of either **Edit** or **Open** command initiates loading of the KOMPAS-3D system. KOMPAS-3D will open the inserted document file source.

If the inserted KOMPAS-document has no connection with the source file, after activation of the **Edit** command window KOMPAS-3D is loaded to the Word window: the Word menus and panels will be replaced with the same of the KOMPAS-3D system. The inserted document will be opened in the KOMPAS window.

After activation of the **Open** command KOMPAS-3D system is loading. The inserted document will be opened in this program.

3. Make the necessary modification in the document.

If you work with a Word document, click on any place not occupied by the embedded document.

If you work in KOMPAS-3D, save the document and exit the session.

The modified KOMPAS-document will appear in the Word window.



We do not recommend to drag the outline frame of the embedded KOMPAS-document during normal work in Word. Such action may cause image distortion in the inserted document.

12.2.2.3. Source File Link Refresh

To make a forced update of the link between inserted KOMPAS-document and a source file, click the right mouse button on the embedded KOMPAS-document and select command **Refresh Link** from the context menu.



To work with links/connections to source files (preset update type, change source, update or break connection), follow prompting messages in the dialog accessible through command **Edit— Links** from the Word menu.

12.2.2.4. Delete Embedded KOMPAS-document

To delete embedded KOMPAS-document:

1. Click on the embedded KOMPAS-document. It will be selected with an overall frame.
2. Press <Delete> key, or select command **Edit— Delete** in the Word menu.

12.3. Attributes

12.3.1. General Information on Attributes

The attribute is additional non-graphical information related to the object or several objects of the drawing, the drawing itself, specification, fragment or fragment in the fragment library.

Attribute values can be used for searching objects or processed by various applications (for example, by the BOM design system, various analyzer programs, etc.).

Different information related to the object (image or document) may be stored in attributes. Depending on the nature of the information, it may be represented either way, i.e., using a certain attribute **structure**.

In KOMPAS-3D, the attribute may have one of the following four structures:

- ▼ Number
- ▼ String of text
- ▼ Table with fixed number of rows
- ▼ Table with arbitrary number of rows.

The same structure may be used to represent different information, i.e., different attributes may have similar structure.

For example, the same object may have several table attributes differing by number of rows and columns, or several text row attributes containing different information about this object.

Information on the attribute structure (and also information on table parameters and cell contents for table attributes) is contained in the attribute **type**. Each attribute type has a **name**. The user can create the attribute types which may be stored in the document (in this case, they are available only when working with this document) or in external **.lat* files — attribute type libraries (in this case, they are available for working with any document).

General procedure of attribute assigning (conferring) to the drawing object or document is as follows:

1. Selecting the attribute type
2. Entering the attribute value.

The **attribute** value is the information stored in the attribute which is assigned to the object. The value of one and the same attribute assigned to different objects may differ.

Therefore, attributes may be viewed as classes (or sections) of information about objects, and attribute values pertain to the information itself related to one or another class (or section). For different objects, this information may be the same or different due to the fact that attribute descriptions (attribute types) are stored separately from the object attributes (attribute values).

12.3.2. Creation of Attribute Types

The procedure of attribute type creation is as follows.

1. Call the command **Service— Libraries of Styles, Layout Types and Title Blocks— Attribute Types....**

The dialog for working with attribute types will appear on the screen (Fig. 12.3.1).

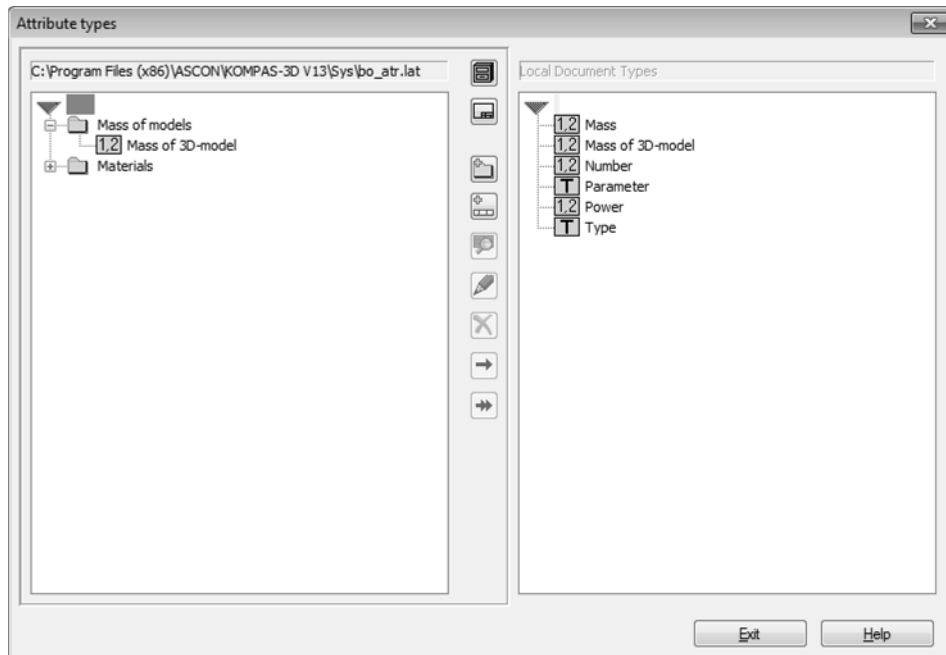


Fig.12.3.1. Dialog to manage attribute types

The dialog has two panels for viewing the list of attribute types. Only one preview window can be active. For example, in Fig. 12.3.1, the right window is active. To activate, click the mouse in the window.

- ▼ Activate Preview window under the heading **Local Document Types** to create a new attribute type in current document. If attribute type libraries are shown in both windows (window headers contain paths to library files), press **Document** button. The list of attribute types of the current document appears in the active window.
 - ▼ Press **Library** button to create a new attribute type in any library. Select the necessary *.lat file in the dialog which appeared on the screen, and press **Open**¹ button. The list of attribute types from a selected library appears in the active preview window. Select the library section to which a new type is to be added.
2. Click the **New Type** button.

The Create Attribute Type dialog will appear (Fig. 12.3.2).

1. You can create your own library for storing attribute types (see Section 12.3.2.2 on p. 2062).

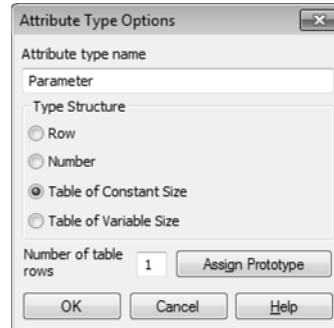


Fig.12.3.2. Dialog to create new attribute type

3. Enter the name of the attribute type created.
4. Enable the option corresponding to the structure of the attribute type created.
5. Click the **Exit** button in the dialog.

This is the last step for the row or number attribute creation: the icon corresponding to the structure of attribute type created along with its name appear in the dialog of the work with attribute types.

Creating a table attribute has its own specifics. They are described in the Section 12.3.2.1.

12.3.2.1. Creation of Table Type Attribute

If the attribute is a table, the attribute type records the number of table columns, and, for each column title and comment, data type, range of data values (if applicable) or the list of allowed values. The table of an attribute is organized so that all cells of its every column contain data of the same type and have the same limitations for the value range. Therefore, an attribute table always has a regular structure.

12.3.2.1.1. Prototype Selection

After enabling the option **Table of Constant Size** or **Table of Variable Size**, **Assign Prototype** button appears in the dialog of attribute type creation. After pressing this button, a set of **Prototype** buttons appears at the bottom of the dialog, and the window of prototype selection appears on the right (Fig.12.3.3).

The description of **Prototype** buttons group is shown in the Table 12.3.1.

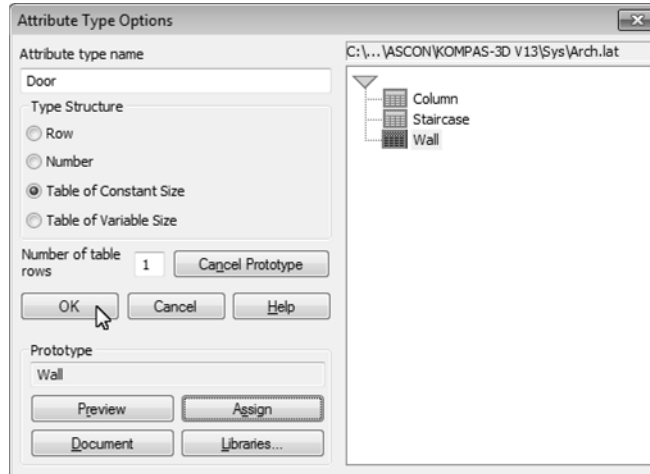


Fig.12.3.3. Selection of table attribute prototype

Tab. 12.3.1. Buttons of **Prototype** group

The	Function
Document	Show attribute types stored in the current document (local attribute types of the document) in the window of prototypes selection.
Libraries...	Showing attribute types stored in external file of *.lat attribute type library in the window of prototypes selection.
Preview	View attribute type selected in the window of prototype selection.
Assign	Assign selected attribute type as a prototype for created one.

12.3.2.1.2. Structure Description

After creation of a table attribute is confirmed (dialog of attribute creation is closed by pressing **OK** button), the structure description dialog of the table attribute appears on the screen (Fig.12.3.4).

A table is displayed in the lower part of the dialog which is used to create or edit a record structure description. Each row contains information on the column generating method in the table attribute. The row in which the cursor is located is considered active.

Control elements of the structure description dialog of the table attribute are shown in the Table12.3.2.

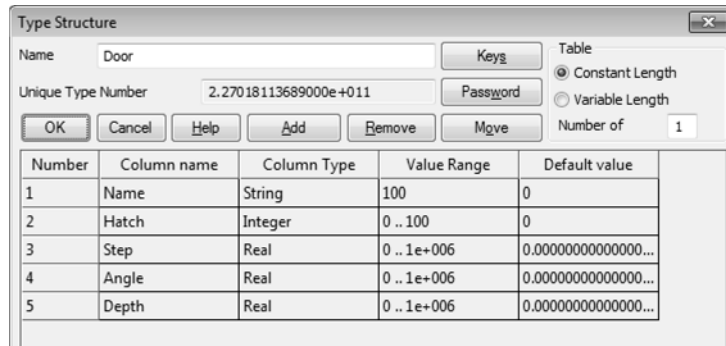


Fig.12.3.4. Dialog to describe the structure of table attribute type

Tab. 12.3.2. Dialog to describe the structure of table attribute type

Name	Description
Name	The field for editing the name of created attribute type.
Unique Type Number	The reference field containing the unique number automatically assigned by the system to the attribute type created. A newly created attribute type has no number. It may be seen during a subsequent viewing, or when editing generated attribute.
Table	Set the table attribute type (table of constant or variable size) using switches. In case of the table of constant size, enter the necessary value in the field Number of Rows .
Number	An automatically assigned column ID number.
Column Name	The field for entering the attribute column name (header).
Column Type	The field for selection of the column data type. To call the dialog of the data type editing, left-click the field, and then press the appeared Change Type button. The dialog of the column data type selection is described in the next section.
Value Range	<p>The field for setting up value range allowed for the column data. To change the range of values, left-click the field, and then press the appeared button.</p> <p>It is necessary to specify the maximum and minimum values for integers and real numbers, as well as the maximum length in characters for the text line*.</p> <p>If the data type in the column is the <i>record</i>, then clicking the button Value Range calls the dialog for generating record structure. See details on record creation on <i>Generating the Record Structure</i> on p. 2062.</p>
Default Value	The field for default value entering in the attribute column.

Tab. 12.3.2. Dialog to describe the structure of table attribute type

Name	Description
Add	The button allowing a new column to be added to the attribute. After pressing this button, a new row located under the row being active at the moment of pressing the button appears in the dialog of the attribute structure description.
Delete	The button allowing to delete a column from the attribute. Pressing this button removes the row being active at the moment of button activation from the dialog of the attribute structure description.
Move	The button allowing to move the column. After pressing this button, specify new position of the active row.
Password	The button allowing to set an access password for the attribute structure (e.g., for protection from unauthorized structure editing).
Keys	The button allowing to set attribute keys. These are used for object sorting by attributes.

* The maximum possible range for integers: from -2147483647 to 2147483647.
 The maximum possible range for real numbers: from -1.7e+307 to 1.7e+307.
 The maximum possible line length: 4,000 characters.

When the structure description is finished, press **OK** button in the dialog. The icon of created table attribute and its name will appear in the dialog of attribute type operation.

Data Type Selection

The dialog of the column data type selection is shown in the Fig.12.3.5.

The control elements of this dialog are represented in Table 12.3.3.

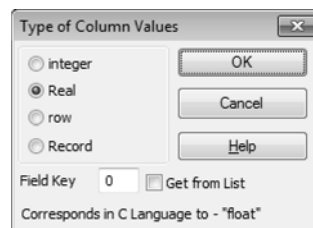


Fig.12.3.5. Dialog of data type selection

Tab. 12.3.3. Dialog of data type selection

Element	Description
Integer, Real, Row, Record	Variants of data type in the attribute column. For detailed information on data type <i>record</i> , see <i>Generating the Record Structure</i> on p. 2062.
Field Key	The field for entering key value for the column. Allowed are integer numbers no greater than 9999.
Get from List	The option allowing to generate the list of values for cells in the column being customized. Later, when the attribute of this type is assigned to the object, entering values into this column will be available only by means of selection from the list (see next section about list generating). If the <i>record</i> data type is selected, this option is unavailable.
Corresponds in C Language to	The info line of the dialog specifies what data type in the C-language corresponds to the selected data type in the column (this information may be needed when developing KOMPAS application libraries).

Generate List of Values for Filling in the Columns

If filling cells with list values is set for a column, the list itself needs to be created.

To do this, left-click the field **Value Range**, and then press the appeared **Show List** button (Fig. 12.3.6).

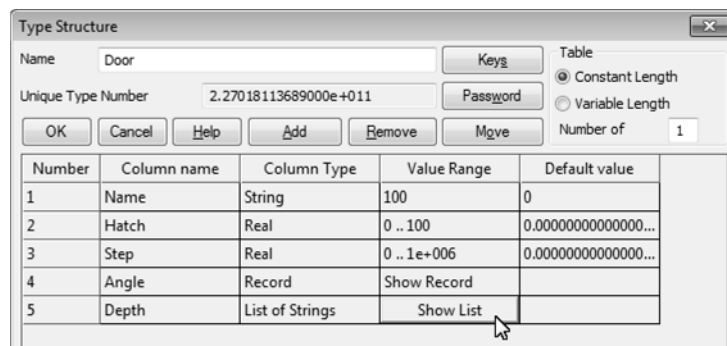


Fig.12.3.6. Calling the value entering dialog

The dialog of the list value entering appears on the screen (Fig. 12.3.7).

In the lower field of the dialog, enter the value from the list, and press **Add** button. This value will be added to the list, i.e. moved to the upper field. The lower field will clear. Generate the list by repeating this procedure.

To delete value, select it and click the **Delete** button.



Fig.12.3.7. Dialog of the list value entering

Generating the Record Structure

The record is a row in another table to which the cell refers.

To generate this table, left-click the field **Value Range**, and then press the appeared **Show Record** button (Fig. 12.3.8).

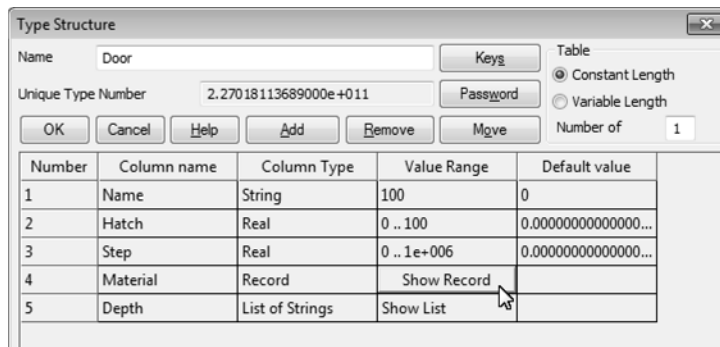


Fig.12.3.8. Calling the dialog of the record structure generating

The dialog of the record structure generating appears on the screen (Fig. 12.3.9).

Generating the record structure is similar to generating the table attribute structure. The difference is that the record cannot be created, i.e., there may be only one nesting level of records.

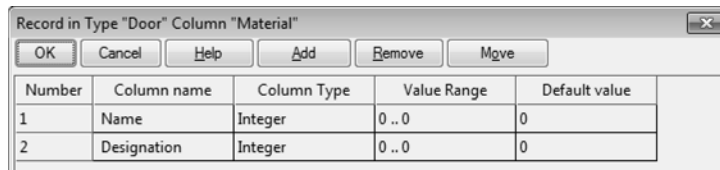


Fig.12.3.9. Dialog of a record structure description

12.3.2.2. Creation of Attribute Type Libraries

To create a new *.lat library, click the **Library** button in the dialog of library operations (Fig. 12.3.1). Enter a non-existing name in the appeared file selection dialog, press **Open** button and confirm the file creation. The active dialog window of attribute type operation will clear, since the newly created library is empty.

If you plan to store a great number of attribute types in the library, it is recommended to divide them into topical groups, or sections; it will simplify future searches for necessary attribute types. To create a section, press **New Section** button and enter section name in the appeared dialog. Generate the library structure by adding new sections.

Then create attribute types in library sections as mentioned above.

Sections, subsections and attribute types will be sorted automatically in alphabetical order.

12.3.2.3. Attribute Type Management

Attribute types are managed from a special dialog called by the **Service — Style Libraries — Attribute Types** command (Fig. 12.3.1).

Functions of **Document**, **Library**, **New Type** and **New Section** buttons are described in previous sections. The remaining buttons are represented in Table 12.3.4.

Tab. 12.3.4. Buttons of the attribute type operation dialog

Name	Description
View	Allows reviewing of selected attribute type characteristics.
Edit	Allows editing of selected attribute type characteristics or selected library section name.
Delete	Allows the deleting of selected attribute type or selected library section (together with attribute types contained in it). Warning! Restoring deleted attribute type or library section is not possible.
Copy	Allows copying of selected attribute type or selected library section [*] . Upon pressing this button, selected attribute type or section appears in the adjacent preview window as well ^{**} .
Move	Allows moving of selected attribute type or selected library section [*] . After this button is pressed, the attribute type or section is deleted from its previous place and moved to an adjacent preview window ^{**} .

* Sections can be copied and moved only between libraries. Only certain attribute types can be copied from the library to the document.

** When copying or moving sections or attributes between libraries, they are pasted into selected library section opened in the adjacent window.

Thus the dialog of attribute type operation allows previewing, editing, and deleting attribute types stored in the current document as well as outside it, in attribute type libraries. This dialog allows editing the library structure as well as copying attribute types moving them between the libraries and documents.

The following should be considered when editing table attribute types:

1. If essential changes are done to the data structure description (moving, deleting or adding columns in a table, changing column data type, changing the range of values in a column, switch-

ing from a table with a fixed number of rows per table to the one with a variable number of rows, etc.), the system will automatically mark the old type as deleted.

2. If changing table attribute type does not affect the data structure (name of the type, column headers in the table, lists of allowed cell values), attribute type is not deleted.



The deletion of number or row attribute types does not affect attributes using these types. However, deleting (manually or as a result of structure editing) a table attribute type makes the editing of attributes using this type impossible.

12.3.3. Use of Attributes

12.3.3.1. Assigning Attributes to Objects and Documents

Attribute assigning to objects means binding objects (images or documents) with attributes of a certain type, and entering attribute value.

12.3.3.1.1. Attribute of one Graphical Object

To assign an attribute to a graphical object (image), do the following.

1. Select objects to which attribute is to be assigned (see Section 3.1.3 on p. 876 on object selection).
2. Call the **Service—Attributes...** command. The dialog of object attribute operation appears on the screen (Fig. 12.3.10).

If the selected object does not have any attributes yet, the list of the existing attributes is empty.

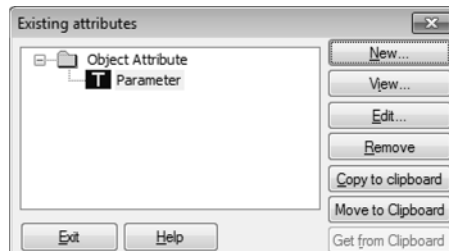


Fig.12.3.10. Dialog to manage object attributes

3. Click **New...** button to call the dialog of attribute type selection (Fig. 12.3.11).
4. Open the necessary list of attribute types by pressing the **Document** or **Libraries...** buttons. If no necessary attribute type is contained in the list, it may be created. To do so, click the **New...** button. The Create Attribute Type dialog will appear (Fig. 12.3.2). Procedure of attribute type creation is described in Section 12.3.2 on p. 2055. The newly created attribute type appears in the list of types.
5. Select the desired attribute type in the list and click the **Select** button. The dialog of the attribute value entering appears on the screen.
6. Enter the value corresponding to attribute type (a number or a text row), or fill in the table.

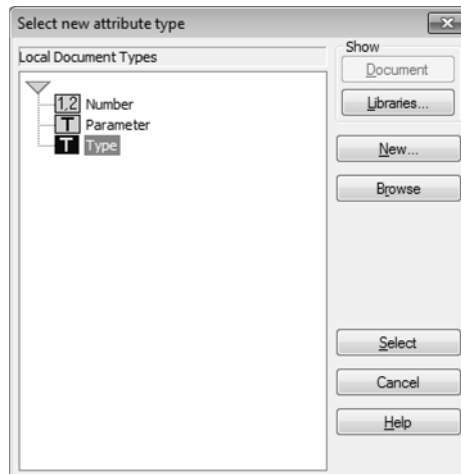


Fig.12.3.11. Dialog to select new attribute type

You can protect the attribute value from unauthorized change by the password. To do so, click **Password**. Enter any combination of characters in the appeared dialog.

The icon corresponding to the structure of selected attribute type and its name will appear in the list of object attributes.

7. To exit the attribute management dialog, click the **Exit** button.

12.3.3.1.2. Attribute of Several Graphical Objects

An attribute can be assigned not only to an individual object, but also to multiple objects simultaneously. In this case a so called group attribute is created. The group attribute may be assigned only to graphical objects; the same attribute cannot be assigned to several documents at the same time.

To assign the group attribute, select necessary objects and follow the same procedure as for assigning an attribute to a single object as a single attribute (see Section 12.3.3.1.1). Fig. 12.3.12 shows the dialog of object attribute operation having single (belonging to one object) attributes as well as group attributes (common to several objects).

Please note that group attributes are contained in a separate folder with the summation symbol image Σ .

If all objects having the same group attributes were selected before calling this dialog, summation symbol will not be shown on the folder containing these objects.



Fig.12.3.12. Object attributes: single and group

Operations with the group attributes have the following specifics.

1. When edited, the value of a group attribute changes for all group objects.
2. Moving a group attribute to the buffer is effected according to the deletion rule mentioned above.
3. A group attribute is deleted only after it is deleted in the last object in the group, or if it is being deleted upon selecting all objects in the group.



If an attempt to delete or edit a value of a group attribute is made, the system will display a warning message.

4. If an object was selected, and you moved a group attribute from it onto clipboard, and later extracted it from the clipboard, that attribute will be saved as a proprietary attribute of the object.

12.3.3.1.3. Document Attribute

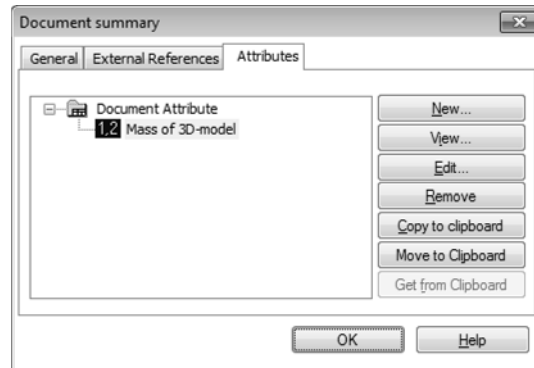
Assigning an attribute to a document is almost identical with assigning it to a graphical object.

To assign an attribute to an active document, do the following.

1. Call the **File — Document Information...** command.
2. Activate **Attributes** tab in the appeared document information dialog (Fig. 12.3.13).
3. Click the **New...** button.
4. Open the necessary list of attribute types by pressing the **Document** or **Libraries...** buttons.
5. Select the desired attribute type in the list and click the **Select** button.

The dialog of the attribute value setting will appear on the screen.

6. Enter the value corresponding to attribute type (a number or a text row), or fill in the table.

Fig.12.3.13. Document Info Dialog **Attributes** tab

12.3.3.2. Copying Attributes between Objects

If you need to assign an attribute to an object whose type coincides with the object type, it is not necessary to create this type again; a prototype copy shall suffice.

You can copy attributes belonging to objects of the same document as well as between objects of different documents, opened in one window of KOMPAS-3D system. You cannot copy attributes belonging to documents opened in different windows of the system.

To copy attributes between objects in KOMPAS-3D, a special clipboard (different from the standard Windows clipboard) is used. The copied attribute value can later be edited.

You may copy attributes between objects of the drawing as follows.

1. Select an object that possesses the attribute intended for copying.
2. Call the **Service —Attributes...** command. The attribute management dialog will appear (Fig.12.3.10).
3. Select the attribute name to be copied in the list, and press **Copy to Clipboard** button. If you click the **Move to Clipboard** button instead, attributes will be copied to the clipboard and then deleted from the source object.



The clipboard may contain only one attribute at a time. Copying or moving the next attribute to the clipboard deletes the previous one.

4. Close the Template use dialog.
5. Select the object to copy the attribute into.
6. Call the **Service —Attributes...** command.
7. Press **Get from Clipboard** button in the appeared dialog.

The attribute contained in the clipboard will be assigned to selected object.

If necessary, edit the contents of the copied attribute.

Copying attributes between documents is also possible. To do this, press the **Copy to Clipboard**, **Move to Clipboard**, and **Get from Clipboard** buttons in **Attributes** tab of the document information dialog (Fig. 12.3.13).

12.3.3.3. Copying Attributes together with Objects

Graphic objects are copied together with attributes. For example, when dragging an object with the mouse keeping the <Ctrl> key pressed, objects copies will have automatically created objects that reproduce the source object's attributes.

You can control copying objects when inserting objects from the clipboard or running symmetry, scale, offset, turn commands, as well as operation of copying by specification, along a curve, along grid, etc.

To do this, use the **Copy Attributes** group of switches on the **Attributes** tab of the Command Properties bar.



Object is copied without attributes when the **Do not copy** switch is activated.



If it is required to create a copy with the same attributes as the attributes of the source object, activate the **Copy** switch.



If a command provides for deleting or retaining source object, attributes are copied if the **Retain source objects** option is selected in the **Mode** group of the Property bar.



If none of the objects selected for copying has attributes, the **Copy Attributes** switches will be inaccessible.

12.3.3.4. Operations with Object Attributes

Operations with attributes of geometric objects are conducted in the dialog (Fig. 12.3.10), called by the **Service— Attributes...** command.

Operations with document attributes are conducted in the **Attributes** tab of the document information dialog (Fig.12.3.13), called by the **File — Document Information...** command.

Functions of **New...**, **Copy to Clipboard**, **Move to Clipboard**, and **Get from Clipboard** buttons are described in previous sections. The remaining buttons are represented in Table12.3.5.

Tab. 12.3.5. Buttons of the attribute operation dialog

Name	Description
Edit	Allows editing attribute value selected in the list. After pressing the button, the dialog of attribute value entering (the same one as when assigning an attribute to the object) appears on the screen (Fig. 12.3.7). If access password was entered upon assigning attribute, the password will be required after pressing Edit button. In this case, the dialog of attribute value entering will appear only after correct password is entered.
View	Allows reviewing of attribute value selected in the list. Upon pressing the button, the dialog of attribute value review will appear on the screen. You can also double-click the name of the attribute value in the list to review it.

Tab. 12.3.5. Buttons of the attribute operation dialog

Name	Description
Delete	Allows deleting attribute selected in the list. Upon pressing the button, request to confirm deleting will appear on the screen. Attribute deleting cannot be canceled. To restore the deleted attribute to the list, it should be assigned to the object again.

When a fragment is inserted from the disk (see Section 3.7.2.1 on p. 1248) or from the library (see Section 12.1.2.2) into the document, the attributes of inserted document, if any, are also transferred to this document.

You may review them in a standard way. They are shown on the **Document Attributes** “branch” in the attribute operation dialog (Fig. 12.3.14). Please note that no document attribute editing is available.

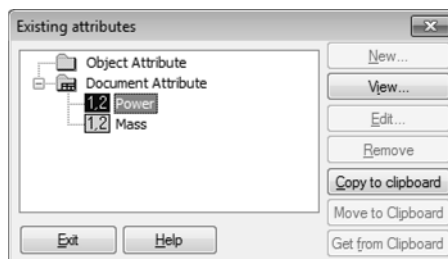


Fig.12.3.14. Review inserted fragment attributes

12.3.3.5. Object Search using Attributes

One of the main purposes of using attributes in drawings and fragments is storing various information related to objects and a fast search using this information. You can quickly select all objects that have the specified attribute type or its value (for example, you can select all machining station images of the same model on a workshop floor plan).

To search and select objects by an attribute type or value, you need to do the following.

1. Call the **Select – By Attributes...** command. The dialog for defining search conditions will appear on the screen (Fig. 12.3.15).

The list of all attributes used in the document, including attributes of fragment insertions, is shown in the upper dialog field.



Fig.12.3.15. Dialog for entering search conditions

2. To use an attribute type for search, select it in the list and click the button with the down arrow. The attribute types selected for the search are displayed in a separate list.

To cancel the use of type when searching, select it in the lower list and press the up arrow button.

You can set up search conditions, i.e. certain attribute values. In this case, not all objects having this specified attribute will be selected, but only those whose attribute values are in accordance with the condition specified.

To set up a search condition, press **Edit Condition** button. The dialog to enter search conditions will appear on the screen. The set of control elements of this dialog depends on the attribute type (row, number, or table); operations with them are described in sections 12.3.3.5.1, 12.3.3.5.2 and 12.3.3.5.3 66.4.3, respectively.

In case you need to delete search condition by attribute value, select the attribute type name for which the condition is set in the lower list and press **Clear Condition** button.

Having set the necessary condition, press **OK** button to start search.

All objects to which at least one of set attributes (if the condition is not set up) or at least one of set attributes whose value satisfies the set condition is assigned will be selected. Different operations may be then performed (i.e., copy, move, etc.).

12.3.3.5.1. Attributes of Number Type

The dialog of search conditions for attributes of number type (Fig. 12.3.16) allows setting of search conditions by values for attributes having integer or real type (as well as search condition by values in integer and real columns of table attributes).

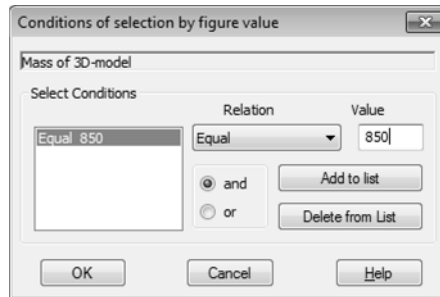


Fig.12.3.16. Dialog to enter search conditions for attributes of the number type

The name of attribute type or column header of the table attribute for which conditions are set is shown at the top of the dialog.

Setting conditions is as follows.

1. Enter the value for condition generating in the **Value** field.
2. Select the necessary selection condition in **Relation** list.
3. Press **Add to List** button to fix condition, i.e., to move it to the list of search conditions at the left of the dialog.
4. Generate necessary number of conditions.
To remove any condition from the list, select it and press **Delete from List** button.
5. Select the type of logical operation, i.e. **and** or **or** to be performed with conditions when this operation is enabled.

12.3.3.5.2. Attributes of Row Type

The dialog of search conditions for attributes of row type (Fig. 12.3.17) allows setting of search conditions by values for attributes of row type (as well as search condition by values in row columns of table attributes).

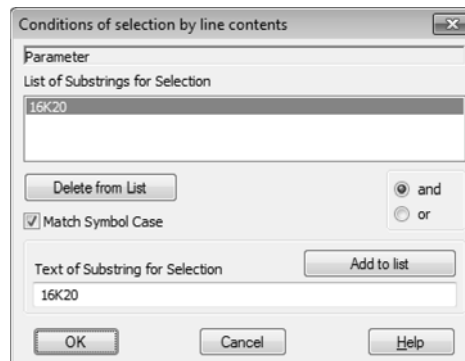


Fig.12.3.17. Dialog to enter search conditions for attributes of the line type

The name of attribute type or column header of the table attribute for which conditions are set is shown at the top of the dialog.

Setting conditions is as follows.

1. Enter substring for condition generating in **Text of Substring for Selection** field.
2. Press **Add to List** button to fix condition, i.e., to move it to the list of substrings at the top of the dialog.
3. Generate necessary number of conditions.
4. To remove any substring from the list, select it and press **Delete from List** button.
5. Select the type of logical operation, i.e. **and** or **or** to be performed with conditions when this operation is enabled.

If case matching shall become necessary for searching, enable **Match Symbol Case** option.

12.3.3.5.3. Attributes of Table Type

The dialog of search conditions entering for the attributes of table type allows setting the search conditions by values in attribute columns of table type.

All fields and options of this dialog as well as table columns, except for **Select Condition**, are provided for information only and cannot be edited.

To set up search condition by any attribute column, click the necessary cell of **Select Condition** column, and press the **Show Condition** button.
















The dialog of search conditions entering for the attributes of row or number type will appear on the screen (its type depends on the value type in selected column). Management of these options is described in Sections 12.3.3.5.1 and 12.3.3.5.2.

If the data type is in the *record* column, click **Value Range** column to set up search conditions, and press **Show Record** button. The record structure dialog will appear on the screen. To set up search conditions by a certain record field, use cells of **Select Condition** column.















Appendix. Terms and Definitions

Appendix I. Symbols in the Document Tree

Tab. 1. Sections of the Document Tree

	Section	Object Types in the Section
	Coordinate Systems	Global and local coordinate systems
	Layout geometry*	Layout geometry
	Components*	Components
	Parts	Branches of the same objects (components or layout geometry).
	Subassemblies	Shown only when the model structure is displayed.
	Library components	
	Standard products	
	Local parts	
	Group of parts	Groups created using the component types or layout geometry. Each group may contain both single objects and branches of similar objects.
	Group of subassemblies	Shown only when grouping is enabled.
	Group of library components	A group of library components may contain standard products.
	Group of local parts	
	Surfaces	Surfaces
	Surface	Operations involved in surface creation
	Bodies	Bodies




Tab. 1. Sections of the Document Tree

	Section	Object Types in the Section
	Body	Operations involved in body creation.
	Solid of pieces	Once the body parts are connected or its integrity is restored, the icon indicating either a body created from parts or body with broken integrity will be replaced with the standard body icon
	Body With Damaged Integrity	
	Mates	Matings
	Construction Geometry	Auxiliary axes, auxiliary planes, control points, attachment points
	Layout Elements	Layout Elements
	Lines and Points	Points, group of points, point arrays, 3D curves, arrays of 3D curves
	Sketches	Sketches
	Measurements	Measurements
	3D macro elements	3D macro elements
	Excluded from Calculation**	Body elements excluded from calculations
	View Inserts	Image inserts from views of other drawings
	Fragment Insertions	Fragment Insertions
	Graphic macro elements	Graphic macro elements












* when editing a component/layout geometry “on the meetings, its icon is colored

** only if there are excluded elements

Tab. 2. Additional Symbols in the Document Tree

Name	Description
(p)	<ul style="list-style-type: none"> ▼ In the model document, marks the current coordinate system in the Document Tree and the current variant in the Variant Tree. ▼ A sign of a current view in a drawing.
(f)	<ul style="list-style-type: none"> ▼ In a model document, indicates a fixed component. ▼ A sign of a background view in a drawing.
(h)	<ul style="list-style-type: none"> ▼ In the document model, this means that the component's dimensions are not nominal but recalculated with respect to the tolerance. ▼ A sign of a hidden view in a drawing.
(+)/(-)	<ul style="list-style-type: none"> ▼ In front of a component name, the (+) sign indicates that the component is fully defined by positioning mates; the (-) sign indicates that the component is not fully defined. ▼ In front of a sketch name, the (+) sign indicates that the sketch is fully defined by parametric limitations; the (-) sign indicates that the component is not fully defined. ▼ In front of a root object, component section, its group or branch, the (+) sign indicates that all included components are fixed and/or fully defined, while the (-) sign is used in all other cases.
(!)	<ul style="list-style-type: none"> ▼ Indicates a sketch that had been re-defined by parametric relations and its inherent limitations.
! Solid of pieces	<ul style="list-style-type: none"> ▼ On part or assembly icon signifies that the model contains solids made up of multiple pieces. ▼ On subassembly icon, signifies that the subassembly contains solids made up of multiple pieces or was divided into pieces as a result of an operation in the assembly. ▼ On the operation icon, this signifies that the operation resulted in a solid made up of parts, or that the initial number of pieces has changed; the number of pieces can be changed by calling the context menu command Change set of parts.
 Error	Error sign. To find out the reason of the error call the context menu command What is wrong?
 Dependency on LCS	It indicates that the object's position and/or orientation are specified with respect to the local coordinate system.
 Excluded from Calculation	Indicates that the object is excluded from calculation.

Tab. 2. Additional Symbols in the Document Tree

Name	Description
	Belongs to macro element Indicates that the object belongs to 3D macro element.
	Unfolded bend Indicates that the sheet feature containing bends is unfolded.
	Simplified Indicates that the Simplified load mode is applied to a component, section, group, or branch of similar components.
	Empty Indicates that the Empty load mode is applied to a component, section, group, or branch of similar components.
	Outline Dimension Indicates that the Dimension load mode is applied to a component, section, group, or branch of similar components.
	Read Only Indicates a component file with the read-only attributes. Shown only when this option was enabled during the access rights configuration (see Section 9.1.4.2 on p. 1694).
	Lock Indicates that the object is unavailable. It is displayed in the following cases: <ul style="list-style-type: none"> ▼ on icons of objects located under End of Model Design (rollback) marker, ▼ on the icon of the object being edited now, and all objects located under it, ▼ on icon of mating if the Empty loading type is applied to one or both its objects, ▼ on icons of operations, component patterns and matings, if at least one of their source components is not available for editing, ▼ on the icons of dependent variant objects that are linked to the respective objects of the source variant.
	Needs to be rebuilt Indicates that rebuilding of a model or associative drawing is required.
	Blue shield Indicates that editing of a component is impossible, but the read-only status can be removed using the Editing — Allowed context menu command.
	Grey shield Indicates that editing of a component is impossible and the read-only status cannot be removed (the read-only status of the component results from applying a user load type to the assembly).
	"Uncut" component Indicates that component is shown not cut in the associated view that contains a cut or a model's section.

Appendix II. System accelerator keys

Tab. 1. **General-system actions**

Keyboard Shortcut	Description
<Enter>, <Spacebar>	Fix (enter) a point. In a graphic document, starts editing mode of the selected object (the cursor must not be on the object).
<Ctrl>+<Enter>	Create an object (finish the operation).
<Esc>	Interrupt command execution, close the menu page or dialog, deselect the selected objects.
<Delete>	Delete all selected objects.
<Ctrl>+<Z>, <Alt>+<BackSpace>	Undo object deletion, view explosion.
<Ctrl>+<+>/<->	Zoom in/out image in the current window in relation to the window center.
<Shift>+<+>/<->	Zoom in/out image in the current window in relation to the cursor.
<F1>	Call the KOMPAS-3D help system (context help).
<Shift>+<F1>	Call the What is it? command.
<Ctrl>+<F4>, <Ctrl>+<W>	Close the active window of the document.
<Alt>+<F4>	Finish work with KOMPAS-3D.

Tab. 2. **Cursor control when entering and editing table text**

Keyboard Shortcut	Description
Cursor control when entering and editing table text	
<Tab>	Move to the next cell.
<Shift>+<Tab>	Move to the previous cell.
<Ctrl>+<←>	Move the cursor one word left inside a cell and then move to the previous cell.
<Ctrl>+<→>	Move the cursor one word right inside a cell and then move to the next cell.

Tab. 3. **Cursor control when entering and editing text**

Keyboard Shortcut	Description
<←>	Move the cursor one position left.
<→>	Move the cursor one position right.
<↓>	Move the cursor one line down.
<↑>	Move the cursor one line up.
<Page Up>	Move the cursor to the upper line of the current window.
<Page Down>	Move the cursor to the bottom line of the current window.
<Home>	Move the cursor to the beginning of a line.
<End>	Move the cursor to the end of a line.
<Ctrl>+<←>	Move the cursor one word left. In a table, you will first move within a cell and then move to the previous cell.
<Ctrl>+<→>	Move the cursor one word right. In tables the movements are performed inside a cell, and then the cursor will shift to the next cell.
<Ctrl>+<↑>	Move the cursor to the beginning of the current paragraph. If the cursor is at the beginning of the paragraph, then move to the beginning of the previous paragraph.
<Ctrl>+<↓>	Move the cursor to the beginning of the next paragraph. From the starting position of the last paragraph the cursor moves to the end of this paragraph.
<Ctrl>+<Home>	Move the cursor to the beginning of the first paragraph of the document or drawing text being edited.
<Ctrl>+<End>	Move the cursor to the end of either the last paragraph of the document or the drawing text being edited.
<Tab>	Move right to the preset tabulation value.
<Enter>	Start the next paragraph.
<Ctrl>+<Enter>	Start the next paragraph on a new page (i.e. insert a hard page break).
<Shift>+<Enter>	Start a new line in the current paragraph (i.e. insert a hard line break).

Tab. 4. **Special actions when entering and editing text**

Keyboard Shortcut	Description
<Delete>	Delete one character to the right of the cursor (the right part of the line will move left). If the cursor is located just before the paragraph end mark, pressing the <Delete> key will merge two paragraphs.
<Ctrl>+<Delete>	Delete one word to the right of the cursor (the right part of the line will move left).
<Backspace>	Delete one character to the left of the cursor (the right part of the line will move left). While deleting the leftmost symbol in the line, the cursor moves to the last position of the previous line.
<Ctrl>+<Backspace>	Delete one word to the left of the cursor (the right part of the line will move left).
<Ctrl>+<Shift>+<Spac e>	Insert a nonbreaking space.
<Ctrl>+<Shift>+<->	Insert a nonbreaking hyphen.
<Insert>	Switch between the insert and replace modes. In the insertion mode the symbol is inserted in the cursor position, drawing apart other symbols. In the replacement mode the entered symbols replace the symbols to the right of the cursor. When the replacement mode is enabled, the displayed cursor becomes thicker to indicate this mode.
<Ctrl>+<Space>	Adjust text parameters to the set style.
<Ctrl>+<Shift>+<←>	Select one word to the left of the cursor.
<Ctrl>+<Shift>+<→>	Select one word to the right of the cursor.
<Shift>+<←>	Select one character to the left of the cursor.
<Shift>+<→>	Select one character to the right of the cursor.
<Shift>+<↑>	Select one line to the left of the cursor.
<Shift>+<↓>	Select one line to the right of the cursor.
<Ctrl>+<Shift>+<R>	Convert selected characters to Cyrillic ones.
<Ctrl>+<Shift>+<L>	Convert selected characters to Roman ones.
<Ctrl>+<Shift>+<D>	Convert selected characters to the lower case.
<Ctrl>+<Shift>+<U>	Convert selected characters to the upper case.

Tab. 4. **Special actions when entering and editing text**

Keyboard Shortcut	Description
<Ctrl>+<F7>	Check spelling in all text objects of the document or in the whole text object that is currently being edited.

Tab. 5. **Window management**

Keyboard Shortcut	Description
<Ctrl>+<F6>, <Ctrl>+<Tab>	Move to the next window.
<Ctrl>+<Shift>+<F6>, <Ctrl>+<Shift>+<Tab>	Move to the previous window.

Tab. 6. **Window display management**

Keyboard Shortcut	Description
<Ctrl>+<G>	Enable/disable grid drawing in the active window.
<Ctrl>+<F9>	Update image in the active window.
<PageUp>	Scroll image one screen up.
<PageDn>	Scroll image one screen down.
<Home>	Scroll image to the top border of the document.
<End>	Scroll image to the bottom border of the document.
<Ctrl>+<PageUp>	Scroll image one screen left.
<Ctrl>+<PageDn>	Scroll image one screen right.
<Ctrl>+<Home>	Scroll image to the left border of the document.
<Ctrl>+<End>	Scroll image to the right border of the document.

Tab. 7. **Cursor position control and snapping**
(Use keys on the auxiliary numeric keypad; the NumLock mode must be enabled (the NumLock LED is on).)

Keyboard Shortcut	Description
<Ctrl>+<0>	Move the cursor to the point (0,0) of the current coordinate system.

Tab. 7. **Cursor position control and snapping**
 (Use keys on the auxiliary numeric keypad;
 the NumLock mode must be enabled (the NumLock LED is on).)

Keyboard Shortcut	Description
<Ctrl>+<Shift>+<5>	Place the cursor at the characteristic element point nearest to it with regard to background views and layers.
<Ctrl>+<5>	Place the cursor at the characteristic element point nearest to it without regard to background views and layers. This key combination is also available when the NumLock mode is disabled. In this case, it allows to place the cursor at the characteristic element point nearest to it with regard to background views and layers.
<Ctrl>+<. >	Place the cursor normally at the nearest point of the nearest element.
<Alt>+<5>	Place the cursor at the intersection point of two objects that is nearest to it.
<Shift>+<5>	Place the cursor at the object middle point that is nearest to it.
<Ctrl>+<1>	Snapping to the nearest element along the diagonal between the negative direction of the OX axis and negative direction of the OY axis of the current coordinate system.
<Ctrl>+<2>, <Ctrl>+<↓ >	Snapping to the nearest element opposite to the direction of the OY axis of the current coordinate system.
<Ctrl>+<3>	Snapping to the nearest element along the diagonal between the positive direction of the OX axis and negative direction of the OY axis of the current coordinate system.
<Ctrl>+<4>, <Ctrl>+<← >	Snapping to the nearest element opposite to the direction of the OX axis of the current coordinate system.
<Ctrl>+<6>, <Ctrl>+<→ >	Snapping to the nearest element along the OX axis of the current coordinate system.
<Ctrl>+<7>	Snapping to the nearest element along the diagonal between the negative direction of the OX axis and positive direction of the OY axis of the current coordinate system.
<Ctrl>+<8>, <Ctrl>+<↑ >	Snapping to the nearest element along the OY axis of the current coordinate system.
<Ctrl>+<9>	Snapping to the nearest element along the diagonal between the positive direction of the OX axis and positive direction of the OY axis of the current coordinate system.

Tab. 8. **Stepwise cursor movement**
 (Use keys on the auxiliary numeric keypad;
 the NumLock mode must be enabled (the NumLock LED is on).)

Keyboard Shortcut	Description
<Ctrl>+<Alt>+<1>	Move the cursor one step along the negative OX direction and negative OY direction of the current coordinate system (CS).
<Ctrl>+<Alt>+<2>	Move the cursor one step along the negative OY direction of the CS.
<Ctrl>+<Alt>+<3>	Move the cursor one step along the positive OX direction and negative OY direction of the CS.
<Ctrl>+<Alt>+<4>	Move the cursor one step along the negative OX direction of the CS.
<Ctrl>+<Alt>+<6>	Move the cursor one step along the positive OX direction of the CS.
<Ctrl>+<Alt>+<7>	Move the cursor one step along the negative OX direction and positive OY direction of the CS.
<Ctrl>+<Alt>+<8>	Move the cursor one step along the positive OY direction of the CS.
<Ctrl>+<Alt>+<9>	Move the cursor one step along the positive OX direction and positive OY direction of the CS.
If the NumLock mode is disabled (the NumLock LED is off), you can move in the following directions:	
<Ctrl>+<Alt>+<2>	Move the cursor one step along the negative OY direction of the CS.
<Ctrl>+<Alt>+<4>	Move the cursor one step along the negative OX direction of the CS.
<Ctrl>+<Alt>+<6>	Move the cursor one step along the positive OX direction of the CS.
<Ctrl>+<Alt>+<8>	Move the cursor one step along the positive OY direction of the CS.
In addition, if no command is called and NumLock mode is disabled, you can use keys <2>, <4>, <6>, <8>, and the arrow keys to move the cursor.	

Tab. 4. **Work with the Property bar and in dialogs**

Keyboard Shortcut	Description
<Tab>	Switch from one control to another in the forward direction.
<Shift>+<Tab>	Switch from one control to another in the backward direction.
<↑>, <↓>	Search values in a list (including a list with a counter). Search options or variants within a group.
<→>, <←>	Search switches in a switch group on the Property Bar. Search options or variants within a group.

Tab. 4. **Work with the Property bar and in dialogs**

Keyboard Shortcut	Description
<Space>	Activating the switch of the Property Bar, which has the focus. Enable/Disable the option or click the button which has the focus. (Focus — selecting a control element. Indicated by a dotted frame around the element.)

Tab. 5. **Model rotation**

Keyboard Shortcut	Description
<Ctrl> + <Shift> + <↑>	Rotate upward in the vertical plane perpendicular to the screen plane.
<Ctrl> + <Shift> + <↓>	Rotate downward in the vertical plane perpendicular to the screen plane.
<Ctrl> + <Shift> + <→>	Rotate rightward in the horizontal plane.
<Ctrl> + <Shift> + <←>	Rotate leftward in the horizontal plane.
<Alt> + <→>	Rotate counterclockwise in the screen plane.
<Alt> + <←>	Rotate clockwise in the screen plane.
<Space> + <↑>	Rotate 90° up in the vertical plane, perpendicular to the screen plane.
<Space> + <↓>	Rotate 90° down in the vertical plane, perpendicular to the screen plane.
<Space> + <→>	Rotate 90° right in the horizontal plane.
<Space> + <←>	Rotate 90° left in the horizontal plane.
<Alt> + <↑>	Rotate 90° clockwise in the screen plane.
<Alt> + <↓>	Rotate 90° counterclockwise in the screen plane.

Tab. 6. **Special actions in preview mode before printing**

Keyboard Shortcut	Description
<Y>	Calls Output Settings Dialog
<Z>	Rotates the selected sheets counterclockwise.

Tab. 6. **Special actions in preview mode before printing**

Keyboard Shortcut	Description
<X>	Rotates the selected sheets clockwise.
<E>	Sets the Printed sheet area selection mode on/off.
<D>	Select Sheets by Marquee — selects sheets by a surrounding (left to right) or cutting (right to left) marquee.
<F>	Calls the Fit Document Scale Dialog.
<C>	Calls Closure and Alignment of Sheets command.
<H>	Calls Put Sheets to Page Nodes command.
<G>	Calls Find Overlapping Sheets command.
<Ctrl> + <F>	Calls Print Filters setup dialog.
<Delete>	Removes the selected sheet from the preview.

Tab. 7. **Miscellaneous**

Keyboard Shortcut	Description
<Shift> + <specified model>	Select the whole assembly component, select the whole part.
<Ctrl>+ <"holding" the selected element with the left mouse button>	Copy the selected assembly component, copy the selected drawing element (fragment).
<Ctrl> + <T>	Start searching objects located 'under the cursor'.
<Ctrl> + <D>	Disable/enable global snaps.
<F3>	Find or replace the next occurrence of the specified text.
<F4>	Repeats the last called command.
<F5>	Rebuild.
<F7>	Enable/disable the mode of rounding parameter values.
<F8>	Enable/disable the orthogonal drawing mode.
<F9>	Show the document in full.
<F11>	Sets displaying variable names in dimensions of sketches and operations on/off.

Tab. 7. **Miscellaneous**

Keyboard Shortcut	Description
<Alt> + <F9>	Show the sketch in full.
<Alt> + <X>	Enable fields of the current cursor coordinates on the Current Status bar.
<Ctrl> + <K>	Change the cursor appearance (large or small).
<Shift> + </>	Enable the Cursor's Current Step field.
<Ctrl> + <A>	Select all objects contained in the current fragment or in the current view of the active drawing.
<Shift> + <Enter>, <Space> + <Enter>	Open the caption editing dialog as part of the selected dimension or designation (the cursor should not be positioned over the selected object). Note that the <Space> + <Enter> key combination is only available in graphic documents.
<Ctrl> + <Shift> + <Enter>	Sets object Autocreation on/off (calls Autocreation command).
<Ctrl> + <Shift> + <Z>	Calls Reselect command.
<Ctrl> + <Shift> + <P>	Calls Remember Status command.
<Ctrl> + <Shift> + <C>	Calls Copy Properties command.
<Ctrl> + <C>, <Ctrl> + <Ins>	Copy selected objects to clipboard.
<Ctrl> + <X>, <Shift> + <Delete>	Cut selected objects to clipboard.
<Ctrl> + <V>, <Shift> + <Ins>	Paste clipboard contents to a document.

Appendix III. Formula Reference Guide

The Guide contains the following formulas:

- ▼ identical conversion:
 - ▼ Fraction properties (Table 1 on p. 2091),
 - ▼ Abridged multiplication formulas (Table 2 on p. 2092),
- ▼ trigonometric formulas:
 - ▼ Adding arguments formulas (Table 3 on p. 2092),
 - ▼ Multiple arguments formulas (Table 4 on p. 2093),
 - ▼ Adding trigonometric functions (Table 5 on p. 2094),
 - ▼ Multiplying trigonometric functions (Table 6 on p. 2094),
 - ▼ Power reducing formulas (Table 7 on p. 2095),
 - ▼ Oblique triangle formulas (Table 8 on p. 2095).

The **Formula** column displays formula's standard view, and the **KOMPAS-3D Syntax** column — their representation in KOMPAS-3D syntax. Text from the **KOMPAS-3D Syntax** column can be copied via the clipboard into Variables Window and Property Bar fields, e.g. for setting curve parameters by law.

Tab. 1. Fraction properties

Formula	KOMPAS-3D Syntax
$\frac{a}{b} = \frac{a \cdot m}{b \cdot m}$	$a/b = (a * m) / (b * m)$
$\frac{a}{b} = \frac{\frac{a}{m}}{\frac{b}{m}}$	$a/b = (a/m) / (b/m)$
$\frac{a}{b} = \frac{a}{c} \cdot \frac{c}{b} = \frac{a \cdot c}{b \cdot c}$	$a/b * c/d = (a * c) / (b * d)$ $(a/b) / (c/d) = (a * d) / (b * c)$
$\frac{a}{b} \cdot \frac{c}{d} = \frac{a \cdot c}{b \cdot d}$	$a/b * c/d = (a * c) / (b * d)$
$\frac{a}{b} \pm \frac{c}{d} = \frac{a \cdot d \pm b \cdot c}{b \cdot d}$	$a/b + c/d = (a * d + b * c) / (b * d)$ $a/b - c/d = (a * d - b * c) / (b * d)$
$\frac{c}{(n+a) \cdot (n+b)} = \left(\frac{1}{n+a} - \frac{1}{n+b} \right) \cdot \frac{c}{b-a}$	$c / ((n+a) * (n+b)) = (1 / (n+a) - 1 / (n+b)) * c / (b-a)$

Tab. 2. Abridged multiplication formulas

Formula	KOMPAS-3D Syntax
$a^2 - b^2 = (a-b) \cdot (a+b)$	$a^2-b^2 = (a-b)*(a+b)$
$(a+b)^2 = a^2 + 2 \cdot a \cdot b + b^2$	$(a+b)^2 = a^2+2*a*b+b^2$
$(a-b)^2 = a^2 - 2 \cdot a \cdot b + b^2$	$(a-b)^2 = a^2-2*a*b+b^2$
$a^3 + b^3 = (a+b) \cdot (a^2 - a \cdot b + b^2)$	$a^3+b^3 = (a+b)*(a^2-a*b+b^2)$
$a^3 - b^3 = (a-b) \cdot (a^2 + a \cdot b + b^2)$	$a^3-b^3 = (a-b)*(a^2+a*b+b^2)$
$(a+b)^3 = a^3 + 3 \cdot a^2 \cdot b + 3 \cdot a \cdot b^2 + b^3$	$(a+b)^3 = a^3+3*a^2*b+3*a*b^2+b^3$
$(a-b)^3 = a^3 - 3 \cdot a^2 \cdot b + 3 \cdot a \cdot b^2 - b^3$	$(a-b)^3 = a^3-3*a^2*b+3*a*b^2-b^3$

Tab. 3. Adding arguments formulas

Formula	KOMPAS-3D Syntax
$\sin(\alpha + \beta) = \sin(\alpha) \cdot \cos(\beta) + \cos(\alpha) \cdot \sin(\beta)$	$\sin(a+b) = \sin(a)*\cos(b)+\cos(a)*\sin(b)$
$\sin(\alpha - \beta) = \sin(\alpha) \cdot \cos(\beta) - \cos(\alpha) \cdot \sin(\beta)$	$\sin(a-b) = \sin(a)*\cos(b)-\cos(a)*\sin(b)$
$\cos(\alpha + \beta) = \cos(\alpha) \cdot \cos(\beta) - \sin(\alpha) \cdot \sin(\beta)$	$\cos(a+b) = \cos(a)*\cos(b)-\sin(a)*\sin(b)$
$\cos(\alpha - \beta) = \cos(\alpha) \cdot \cos(\beta) + \sin(\alpha) \cdot \sin(\beta)$	$\cos(a-b) = \cos(a)*\cos(b)+\sin(a)*\sin(b)$
$\operatorname{tg}(\alpha + \beta) = \frac{\operatorname{tg}(\alpha) + \operatorname{tg}(\beta)}{1 - \operatorname{tg}(\alpha) \cdot \operatorname{tg}(\beta)}$	$\tan(a+b) = (\tan(a)+\tan(b))/(1-\tan(a)*\tan(b))$

Tab. 3. Adding arguments formulas

Formula	KOMPAS-3D Syntax
$\operatorname{tg}(\alpha - \beta) = \frac{\operatorname{tg}(\alpha) - \operatorname{tg}(\beta)}{1 + \operatorname{tg}(\alpha) \cdot \operatorname{tg}(\beta)}$	$\tan(a-b) = (\tan(a) - \tan(b)) / (1 + \tan(a) \cdot \tan(b))$

Tab. 4. Multiple arguments formulas

Formula	KOMPAS-3D Syntax
$\sin(2 \cdot \alpha) = 2 \cdot \sin(\alpha) \cdot \cos(\alpha)$	$\sin(2^*a) = 2^*\sin(a)^*\cos(a)$
$\sin(2 \cdot \alpha) = \frac{2 \cdot \operatorname{tg}(\alpha)}{1 + \operatorname{tg}(\alpha)^2}$	$\sin(2^*a) = (2^*\tan(a)) / (1 + \tan(a)^2)$
$\cos(2 \cdot \alpha) = \cos(\alpha)^2 - \sin(\alpha)^2$	$\cos(2^*a) = \cos(a)^2 - \sin(a)^2$
$\cos(2 \cdot \alpha) = 2 \cdot \cos(\alpha)^2 - 1$	$\cos(2^*a) = 2^*\cos(a)^2 - 1$
$\cos(2 \cdot \alpha) = 1 - 2 \cdot \sin(\alpha)^2$	$\cos(2^*a) = 1 - 2^*\sin(a)^2$
$\operatorname{tg}(2 \cdot \alpha) = \frac{2 \cdot \operatorname{tg}(\alpha)}{1 - \operatorname{tg}(\alpha)^2}$	$\tan(2^*a) = (2^*\tan(a)) / (1 - \tan(a)^2)$
$\sin(3 \cdot \alpha) = 3 \cdot \sin(\alpha) - 4 \cdot \sin(\alpha)^3$	$\sin(3^*a) = 3^*\sin(a) - 4^*\sin(a)^3$
$\cos(3 \cdot \alpha) = 4 \cdot \cos(\alpha)^3 - 3 \cdot \cos(\alpha)$	$\cos(3^*a) = 4^*\cos(a)^3 - 3^*\cos(a)$
$\operatorname{tg}(3 \cdot \alpha) = \frac{3 \cdot \operatorname{tg}(\alpha) - \operatorname{tg}(\alpha)^3}{1 - 3 \cdot \operatorname{tg}(\alpha)^2}$	$\tan(3^*a) = (3^*\tan(a) - \tan(a)^3) / (1 - 3^*\tan(a)^2)$

Tab. 5. Adding trigonometric functions

Formula	KOMPAS-3D Syntax
$\sin(\alpha) + \sin(\beta) = 2 \cdot \sin\left(\frac{\alpha + \beta}{2}\right) \cdot \cos\left(\frac{\alpha - \beta}{2}\right)$	$\sin(a) + \sin(b) = 2 * \sin((a+b)/2) * \cos((a-b)/2)$
$\sin(\alpha) - \sin(\beta) = 2 \cdot \sin\left(\frac{\alpha - \beta}{2}\right) \cdot \cos\left(\frac{\alpha + \beta}{2}\right)$	$\sin(a) - \sin(b) = 2 * \sin((a-b)/2) * \cos((a+b)/2)$
$\cos(\alpha) + \cos(\beta) = 2 \cdot \cos\left(\frac{\alpha + \beta}{2}\right) \cdot \cos\left(\frac{\alpha - \beta}{2}\right)$	$\cos(a) + \cos(b) = 2 * \cos((a+b)/2) * \cos((a-b)/2)$
$\cos(\alpha) - \cos(\beta) = -2 \cdot \sin\left(\frac{\alpha + \beta}{2}\right) \cdot \sin\left(\frac{\alpha - \beta}{2}\right)$	$\cos(a) - \cos(b) = -2 * \sin((a+b)/2) * \sin((a-b)/2)$
$\operatorname{tg}(\alpha) + \operatorname{tg}(\beta) = \frac{\sin(\alpha + \beta)}{\cos(\alpha) \cdot \cos(\beta)}$	$\tan(a) + \tan(b) = \sin(a+b) / (\cos(a) * \cos(b))$
$\operatorname{tg}(\alpha) - \operatorname{tg}(\beta) = \frac{\sin(\alpha - \beta)}{\cos(\alpha) \cdot \cos(\beta)}$	$\tan(a) - \tan(b) = \sin(a-b) / (\cos(a) * \cos(b))$

Tab. 6. Multiplying trigonometric functions

Formula	KOMPAS-3D Syntax
$\sin(\alpha) \cdot \sin(\beta) = \frac{1}{2} \cdot (\cos(\alpha - \beta) - \cos(\alpha + \beta))$	$\sin(a) * \sin(b) = 1/2 * (\cos(a-b) - \cos(a+b))$
$\cos(\alpha) \cdot \cos(\beta) = \frac{1}{2} \cdot (\cos(\alpha - \beta) + \cos(\alpha + \beta))$	$\cos(a) * \cos(b) = 1/2 * (\cos(a-b) + \cos(a+b))$
$\sin(\alpha) \cdot \cos(\beta) = \frac{1}{2} \cdot (\sin(\alpha - \beta) + \sin(\alpha + \beta))$	$\sin(a) * \cos(b) = 1/2 * (\sin(a-b) + \sin(a+b))$

Tab. 7. Power reducing formulas

Formula	KOMPAS-3D Syntax
$\cos(\alpha)^2 = \frac{1 + \cos(2 \cdot \alpha)}{2}$	$\cos(a)^2 = (1 + \cos(2 \cdot a))/2$
$\sin(\alpha)^2 = \frac{1 - \cos(2 \cdot \alpha)}{2}$	$\sin(a)^2 = (1 - \cos(2 \cdot a))/2$
$\cos(\alpha)^3 = \frac{3 \cdot \cos(\alpha) + \cos(3 \cdot \alpha)}{4}$	$\cos(a)^3 = (3 \cdot \cos(a) + \cos(3 \cdot a))/4$
$\sin(\alpha)^3 = \frac{3 \cdot \sin(\alpha) - \sin(3 \cdot \alpha)}{4}$	$\sin(a)^3 = (3 \cdot \sin(a) - \sin(3 \cdot a))/4$

Tab. 8. Oblique triangle formulas

Formula	KOMPAS-3D Syntax
$\frac{L1}{\sin(\alpha)} = \frac{L2}{\sin(\beta)} = \frac{L3}{\sin(\gamma)} = 2 \cdot R$	$2 \cdot R = L1/\sin(a)$ $2 \cdot R = L2/\sin(b)$ $2 \cdot R = L3/\sin(c)$
$L3^2 = L1^2 + L2^2 - 2 \cdot L1 \cdot L2 \cdot \cos(\gamma)$	$L3^2 = L1^2 + L2^2 - 2 \cdot L1 \cdot L2 \cdot \cos(c)$
$\frac{L1 - L2}{L1 + L2} = \frac{\operatorname{tg}\left(\frac{\alpha - \beta}{2}\right)}{\operatorname{tg}\left(\frac{\alpha + \beta}{2}\right)}$	$(L1 - L2)/(L1 + L2) = \tan((a - b)/2)/\tan((a + b)/2)$
$R = \frac{p}{4 \cdot \cos\left(\frac{\alpha}{2}\right) \cdot \cos\left(\frac{\beta}{2}\right) \cdot \cos\left(\frac{\gamma}{2}\right)}$	$R = p/(4 \cdot \cos(a/2) \cdot \cos(b/2) \cdot \cos(c/2))$
$r = 4 \cdot R \cdot \sin\left(\frac{\alpha}{2}\right) \cdot \sin\left(\frac{\beta}{2}\right) \cdot \sin\left(\frac{\gamma}{2}\right)$	$r = 4 \cdot R \cdot \sin(a/2) \cdot \sin(b/2) \cdot \sin(c/2)$

Tab. 8. Oblique triangle formulas

Formula	KOMPAS-3D Syntax
$S = \frac{1}{2} \cdot L1 \cdot L2 \cdot \sin(\gamma)$	S = 1/2*L1*L2*sin(c)
$S = 2 \cdot R^2 \cdot \sin(\alpha) \cdot \sin(\beta) \cdot \sin(\gamma)$	S = 2*R^2*sin(a)*sin(b)*sin(c)

Appendix IV. Operators, functions and constants

Tab. 1. Arithmetic operators

Designation	Description
()	operator parentheses
+	add or unary plus
-	subtract or unary minus
*	multiply
/	divide
%	integer division (left associative [*])
^	raise to a degree (right associative ^{**})
=	equal

* In case of equal operators' priorities, the expression's result is calculated **from left to right**, i.e. $a\%b\%c = (a\%b)\%c$.

** In case of equal operators' priorities, the expression's result is calculated **from right to left**, i.e. $a^b^c = a^{(b^c)}$.

Tab. 2. Logical operators

Designation	Description
==	identical
!=	not identical
>	greater than
<	less than
>=	greater than or equal to
<=	less than or equal to
!	logical negation
&&	logical AND
	logical OR
?:	a logical expression of type $a ? b : c$ (if a is true (not equal to 0), then b, else c, where a, b and c can be expressions)

Tab. 3. Functions

Designation	Description
sin	Sine (radian argument)
cos	cosine (radian argument)
tan	tangent (radian argument)
atan	arc tangent (radian result)
acos	arc cosine (radian result)
asin	arc sine (radian result)
sind	sine (degrees argument)
cosd	cosine (degrees argument)
tand	tangent (degrees argument)
atand	arc tangent (degrees result)
acosd	arc cosine (degrees result)
asind	arc sine (degrees result)
deg	convert radians to degrees
rad	convert degrees to radians
sqrt	square root
exp	exponential
ln	natural logarithm
lg	common logarithm
abs	absolute value
ceil	rounding off to a larger integer
floor	rounding off to a smaller integer
round	rounding off to a nearest integer

Tab. 4. Constants

Designation	Value	Description
M_FI	1.6180339887499	φ is an irrational number of the golden section

Tab. 4. Constants

Designation	Value	Description
M_E	2.71828182845904523536	e is the base of a natural logarithm
M_PI	3.14159265358979323846	π is the ratio of circumference to diameter
M_PI_2	1.57079632679489661923	$\pi/2$
M_PI_4	0.785398163397448309616	$\pi/4$
M_PI2	6.28318530717959	2π
M_SQRT2	1.41421356237309504880	Square root of 2
M_RADDEG	57.29577951308	Ratio of converting radians to degrees ($180/\pi$)
M_DEGRAD	0.01745329251994	Ratio of converting degrees to radians ($\pi/180$)
FLT_EPS	1.19209290E-07	The difference between two numbers, at which the numbers are considered equal

Appendix V.KOMPAS-3D special characters

I. Creating Dimensions

1	°	Degree	6	≤	Less Than or Equal to
2	∅	Diameter	7	≥	Greater Than or Equal to
3	±	Plus-minus	80	abcd	Text Framed
82	∓	Minus-plus	83	$\sqrt{\text{abcd}}$	Root Face Size
4	×	Multiply	95	<u>abcd</u>	Overlined Text
14	□	Square	96	<u>abcd</u>	Underlined Text
77	R	Radius	169	abcd	Strikethrough Text
81	M	Metric thread	97	$\overbrace{\text{abcd}}$	Text Bailed
90	○	Sphere	98	$AB\sqrt{CD}EF$	Square Root
68	&	Symbol &	99	$AB\sqrt[3]{CD}EF$	Cubic Root
212	~	Symbol ~	210	N ^o	Number
5	≈	Approximately Equal			

I.I Asterisk with

100	*	Asterisk	184	* ¹⁵	Asterisk with 15
8	**	Double Asterisk	185	* ¹⁶	Asterisk with 16
101	* ¹	Asterisk with 1	186	* ¹⁷	Asterisk with 17
102	* ²	Asterisk with 2	187	* ¹⁸	Asterisk with 18
103	* ³	Asterisk with 3	188	* ¹⁹	Asterisk with 19
104	* ⁴	Asterisk with 4	189	* ²⁰	Asterisk with 20
105	* ⁵	Asterisk with 5	190	* ²¹	Asterisk with 21
106	* ⁶	Asterisk with 6	191	* ²²	Asterisk with 22
107	* ⁷	Asterisk with 7	192	* ²³	Asterisk with 23
108	* ⁸	Asterisk with 8	193	* ²⁴	Asterisk with 24
109	* ⁹	Asterisk with 9	194	* ²⁵	Asterisk with 25
110	* ¹⁰	Asterisk with 10	195	* ²⁶	Asterisk with 26
180	* ¹¹	Asterisk with 11	196	* ²⁷	Asterisk with 27
181	* ¹²	Asterisk with 12	197	* ²⁸	Asterisk with 28

182	^{*13}	Asterisk with 13	198	^{*29}	Asterisk with 29
183	^{*14}	Asterisk with 14	199	^{*30}	Asterisk with 30

I.II Superfix

200	⁰	Superfix 0	204	⁴	Superfix 4
125	⁺	Superfix +	205	⁵	Superfix 5
127	[,]	Superfix ,	206	⁶	Superfix 6
128	¹	Superfix 1	207	⁷	Superfix 7
129	²	Superfix 2	208	⁸	Superfix 8
203	³	Superfix 3	209	⁹	Superfix 9

I.III Subscript

111	₁	Subscript 1	117	₇	Subscript 7
112	₂	Subscript 2	118	₈	Subscript 8
113	₃	Subscript 3	119	₉	Subscript 9
114	₄	Subscript 4	120	₀	Subscript 0
115	₅	Subscript 5	126	₋	Subscript -
116	₆	Subscript 6			


II. Angles, Tapers, Conicity

76		Taper (Left)	13		Conicity (Right)
11		Taper (Right)	9		Angle (Left)
12		Conicity (Left)	69		Angle (Right)





III. Geometric and Location Tolerance

30		Dependent Tolerance	32		Independent Set Tolerance
31		Illegal Tolerance	220		Tolerance for Soft Parts

III.I Geometric Tolerance

16		Straightness Tolerance	23		Cylindricity Tolerance
17		Flatness Tolerance	18		Longitudinal Section Profile Tolerance
22		Circularity Tolerance			

III.II Location Tolerance

24		Parallelism Tolerance	19		Symmetry Tolerance
25		Perpendicularity Tolerance	27		Positional Tolerance

10		Slope Tolerance	33		Axes Intersection Tolerance
26		Coaxial Tolerance			
III.III Cumulative Geometric and Position Tolerance					
28		Run-out Tolerance	20		Preset Profile Geometric Tolerance
29		Total Run-out Tolerance	21		Preset Surface Geometric Tolerance
IV. Roughness Directions					
70		Parallel	74		Circular
71		Perpendicular	75		Radial
72		Intersecting	170		Dotted
73		Arbitrary			
V. Welded Connections					
34		Cathetus Size Symbol	40		Unlocked Line Weld
35		Remove Weld Reinforcement (front)	148		Field Weld Symbol
36		Remove Weld Reinforcement (Reverse)	121		Welding Seam Symbol
37		Finished Cold Laps with Graded Taper (Front)	122		Welding Seam Symbol. Along Contour
38		Finished Cold Laps With Graded Taper (Reverse)	123		Welding Seam Symbol. Mounting Weld
15		Intermittent Weld or Dotted Staggered Weld (1)	124		Welding Seam Symbol. Mounting seam by contour.
39		Intermittent Weld or Dotted Staggered Weld (2)			
V.I ISO 2553:1992 Welded Seams					
1001		Flanged Butt Weld between Plates	1013		Seam Resistance Welding
1002		Square Butt Weld	1014		Flanged Butt Weld between Plates Mirror Symbol
1003		V Butt Weld	1015		V Butt Weld Mirror Symbol
1004		Single-V Butt Weld	1016		Single-V Butt Weld. Mirror Symbol

1005		V Butt Weld with Wide Root Face	1017		V Butt Weld with Wide Root Face Mirror Symbol
1006		Single-bevel Butt Weld with Wide Root Face	1018		Single-bevel Butt Weld with Wide Root Face Mirror Symbol
1007		U-butt Weld	1019		U-butt Weld Mirror Symbol
1008		Single-J Butt Weld	1020		Single-J Butt Weld Mirror Symbol
1009		Back Weld	1021		Back Weld Mirror Symbol
1010		Fillet Weld	1022		Fillet Weld. Mirror Symbol
1011		Plug Weld	1023		Plug Weld Mirror Symbol
1012		Spot Weld			

V.II Other Marks

84		Cathetus Size Symbol (inverted 1)	86		Cathetus Size Symbol (inverted 3)
85		Cathetus Size Symbol (inverted 2)			

VI. View, Cut, Section Symbols

63		Rotated	64		Reoriented
----	--	---------	----	--	------------

VII. Surface Finish Symbols

171		Top. Meas. GOST No. 3 2.309-73	211		Top w/o parameters
172		Top. With Material Removal. Meas. GOST No. 3 2.309-73	91		Top. Without Material Removal
78		Top	92		Lower. Without Material Removal
79		Lower	87		Surface finish; Top. Along Contour
93		Top. With Material Removal.	88		Surface finish; Top. With Material Removal. Along Contour
94		Lower. With Material Removal.	89		Surface finish; Top. Without Material Removal Along Contour

VIII. Special Characters for 4.x-Compatibility

VIII.I Roman Numerals

41	I	Roman 1	46	VI	Roman 6
42	II	Roman 2	47	VII	Roman 7
43	III	Roman 3	48	VIII	Roman 8
44	IV	Roman 4	49	IX	Roman 9
45	V	Roman 5	50	X	Roman 10


























VIII.II Predefined Surface Finish Symbols

65		Rz40	67		Rz20
66		Rz80			

VIII.III Greek Alphabet

51	α	Lowercase Alpha	57	Σ	Upper-case sigma
52	β	Lowercase Beta	58	σ	Lowercase sigma
53	γ	Lowercase Gamma	59	τ	Lowercase Tau
54	Δ	Upper-case Delta	60	φ	Lowercase Phi
55	δ	Lowercase Delta	61	Ω	Upper-case Omega
56	π	Upper-case Pi	62	ω	Lowercase Omega

IX. Structural Sections Notation

130		Circle	143		Channel
131		Square/Rectangle	144		Bulb-tee
132		Hexagonal Section	145		Bulb-angle
133		Triangular Section	146		Equal-flange C-type Section
134		Segmented Profile	147		Equal-flange Zee Beam
135		Trapezoidal Section	213		Unequal Channel
136		Oval Design of Section	214		Unequal Angle
137		Strip Design of Section	215		Twin Equal Angles
138		Flat-bulb	216		Twin Unequal Angles
139		Double Flat-bulb	217		Alternate Angles
140		Angle	218		2-Channel Box
141		T section	219		Twin Channels
142		Double T Section			

X. Revision Numbers

149	①	Revision No. 1	159	⑪	Revision No. 11
150	②	Revision No. 2	160	⑫	Revision No. 12
151	③	Revision No. 3	161	⑬	Revision No. 13
152	④	Revision No. 4	162	⑭	Revision No. 14
153	⑤	Revision No. 5	163	⑮	Revision No. 15
154	⑥	Revision No. 6	164	⑯	Revision No. 16
155	⑦	Revision No. 7	165	⑰	Revision No. 17
156	⑧	Revision No. 8	166	⑱	Revision No. 18
157	⑨	Revision No. 9	167	⑲	Revision No. 19
158	⑩	Revision No. 10	168	⑳	Revision No. 20

XI. Special Characters for Engraving

173	<input type="checkbox"/>	Character Cell 12.0 for characters 2, 3,...	177	<input type="checkbox"/>	Character Cell 17.6 for characters D, M,...
174	<input type="checkbox"/>	Character 14.4 for characters B, V,...	178	<input type="checkbox"/>	Character Cell for characters S, Y,...
175	<input type="checkbox"/>	Character 15.2 for characters I, N, ...	179	<input type="checkbox"/>	Character Cell 20.8 for characters W, U, ...
176	<input type="checkbox"/>	Character Cell 16.0 for characters A, O, ...			

XII. Graphic Notation Conventions in Charts. GOST 2.751-73

201		Twisted Pair	202		H-type Cable
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Appendix VI. Data file

Data File is a file containing numerical values that can be used as parameters of points or variables of array instances.

Data files can have the following formats and, respectively, name extensions:

- ▼ Text, *txt*,
- ▼ Text table, *csv*,
- ▼ MS Excel electronic spreadsheet, *xls*,
- ▼ Open Office spreadsheet, *ods* (in order to save in this format, Open Office should be installed on the PC).



We recommend to close the *.xls file in MS Excel before starting the data transmission from this file to KOMPAS-3D.

When working in KOMPAS-3D, you can both save data into a file and read data from a file.

Saving to a file

Saving data to a file is supported for the following types of objects:

- ▼ spline and polyline (if working with the parameter table, refer to Section 2.5.4.8.1 on p. 376),
- ▼ spline form (if setting up a grid, refer to Section 2.6.3.3.2 on p. 502),
- ▼ arrays (for managing instance variables, see Section 2.7.2 on p. 519; for specifying positions of array instances using a table, see Section 2.7.7.2 on p. 568).

Parameters of each object (a point or an array instance) must be located in a separate line. For a point, maximum number of parameters in a line is four, and for an instance, it is unlimited. The parameter values are delimited by means of a space or several spaces whereas decimal digits are separated by means of a point.

Coordinates of points are measured in the absolute coordinate system.

The number of decimal points in the values in the file depends on the setting made in the number scheme setup dialog (see Section 9.1.2.2 on p. 1668).

For a spline or a polyline, the data lines in the file are grouped together.

In case of the spline form, the strings are combined into several groups. Each group contains pole parameters of one spline form grid row and is separated from the next group by an empty line. The number of groups is equal to the number of grid rows.

Reading from file

Reading from a file is possible for the following objects:

- ▼ surface on grid of points (see Section 2.6.1.6.2 on p. 451),
- ▼ surface on point layer (see Section 2.6.1.7.2 on p. 456),
- ▼ spline and polyline (see Section 2.5.4.8 on p. 376),
- ▼ point group from a file (see Section 2.5.3.4 on p. 358),

- ▼ imported surface (see Section 2.6.1.1 on p. 444),
- ▼ arrays (see Section 2.7 on p. 509).
To ensure correct reading of a data file, the file shall meet the following requirements.
- ▼ Parameters of each object (a point or an array instance) must be located in a separate line.
- ▼ No empty parameter value fields are allowed.
- ▼ For a point, maximum number of parameters in a line is four. The list and sequence of parameters depending on the object type are given in Table 1.
- ▼ For an instance, the maximum number of parameters in a line is unlimited.
- ▼ The parameter values in text files are delimited by means of a space or several spaces whereas decimal digits are separated by means of a point.
- ▼ Angle values are written in the decimal number system format.
- ▼ File rows beginning with “#” are treated as comments.
Additional requirements for data files for surfaces along a point grid:
 - ▼ Rows of points are described by groups of rows; the number of rows in groups is identical,
 - ▼ Groups are separated with an empty row or a row containing any integer value (usually the row number) or one of the following symbols: plus (+), minus (-), asterisk (*); several continuous group separators are treated as one separator.



The data file for a surface built along a point grid may contain one group or a fractional number of groups. Should this be the case, specify or construct the missing points after reading the file to enable generation of a surface.

Tab. 1. Point parameters required for correct data reading from file

Object type	Object Point Parameters Stored in a File	Sequence of Parameters in a Row
Surface by Point Grid, Surface by Point Formation, Imported Surface	Rectangular Coordinates	X Y Z
Spline by Poles	Rectangular Coordinates and Weights	X Y Z weight
Polyline	Rectangular Coordinates and Spherical Radii at Vertices	X Y Z radius
Group of points from file, array by table	Rectangular, Cylindrical or Spherical Coordinates*	X Y Z, R A Z or R A B

* The option for interpretation of the data recorded in a file is selected by the user while setting the parameters of the operation.

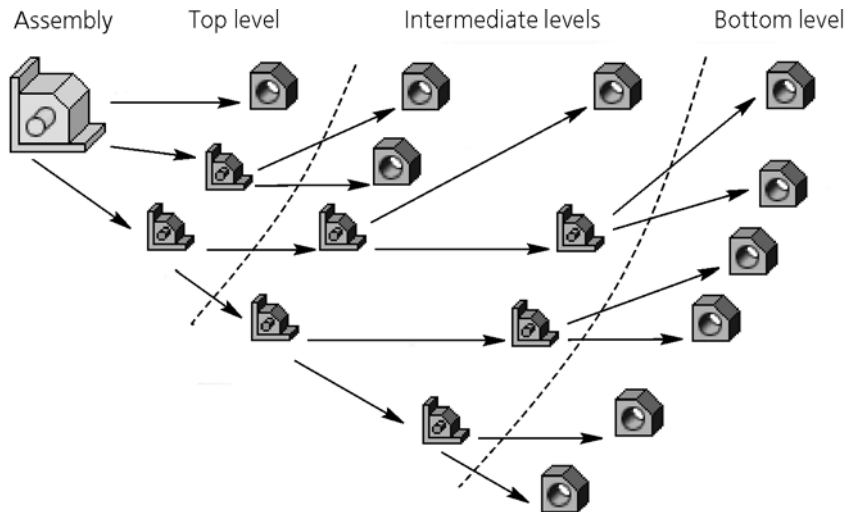
Appendix VII. Methods of assembly designing. Group work

General Information

The following approaches are used for designing assemblies:

- ▼ Top-down designing,
- ▼ Down-top designing.

Top-down designing.

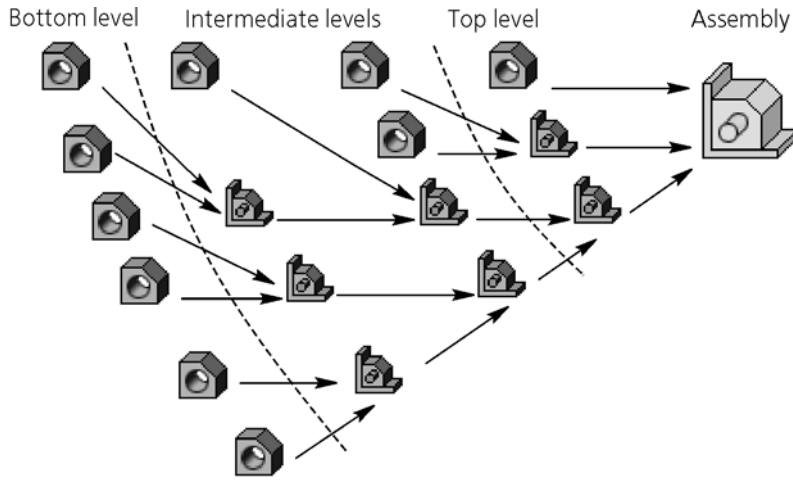


Workflow of top-down designing

Top-down designing is based on the following principles:

- ▼ the designing starts at the top level and ends at the lower levels;
- ▼ components are mainly created in the assembly's context.

Down-top designing.



Workflow of down-top designing

Down-top designing is based on the following principles:

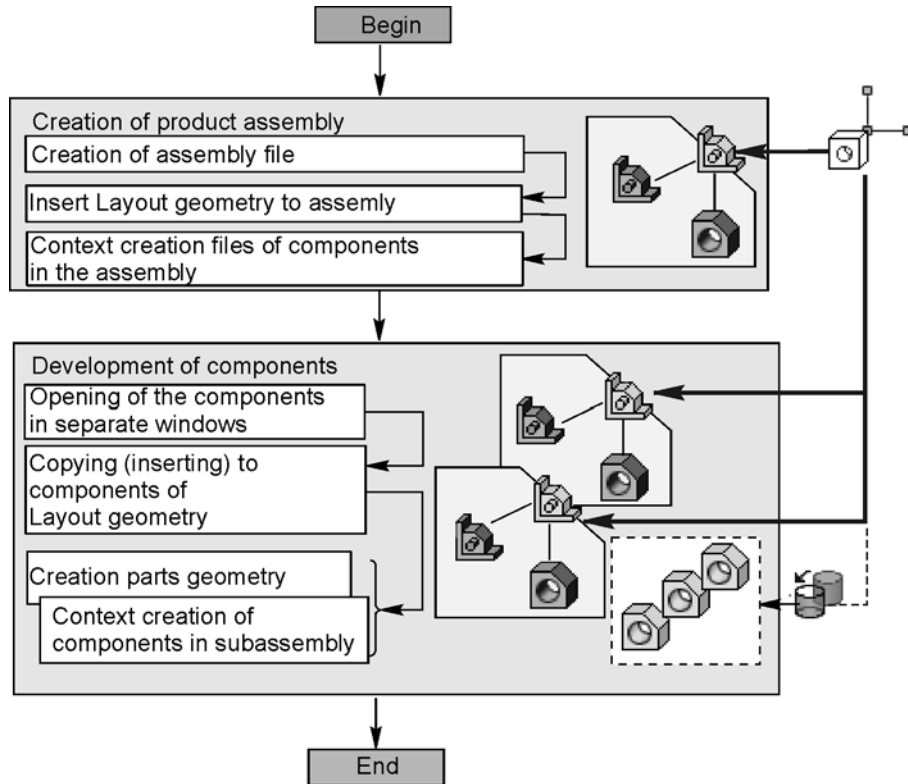
- ▼ the designing starts at the lower levels and ends at the top level;
- ▼ components are mainly created in separate windows outside the assembly's context;
- ▼ the completed components are placed into the assembly (subassembly).

Based on the above approaches, the following designing methods are implemented in the system:

- ▼ **Top-down with pre-packaging.**
- ▼ **Top-down with conversion of bodies into components.**
- ▼ **Down-top with pre-packaging.**
- ▼ **Down-top with arrangement of components.**

These methods are intended to help you choose the best designing scenario. In real designing projects, multiple methods are often combined.

The “top-down with pre-packaging” method.



Description of the top-down design with pre-packaging

The method is based on a sequential, from the top level down to the lower ones, creation of components in the contexts of models they are kept in. Each created component is then developed in a separate window. The parts are designed according to the copies of the objects of the composition geometry, or on the insertions of the composition geometry (when in assemblies).

This method is used in the following cases:

- ▼ the components themselves and their design are not known at the initial phase;
- ▼ most of the parts and nodes are designed for the first time.

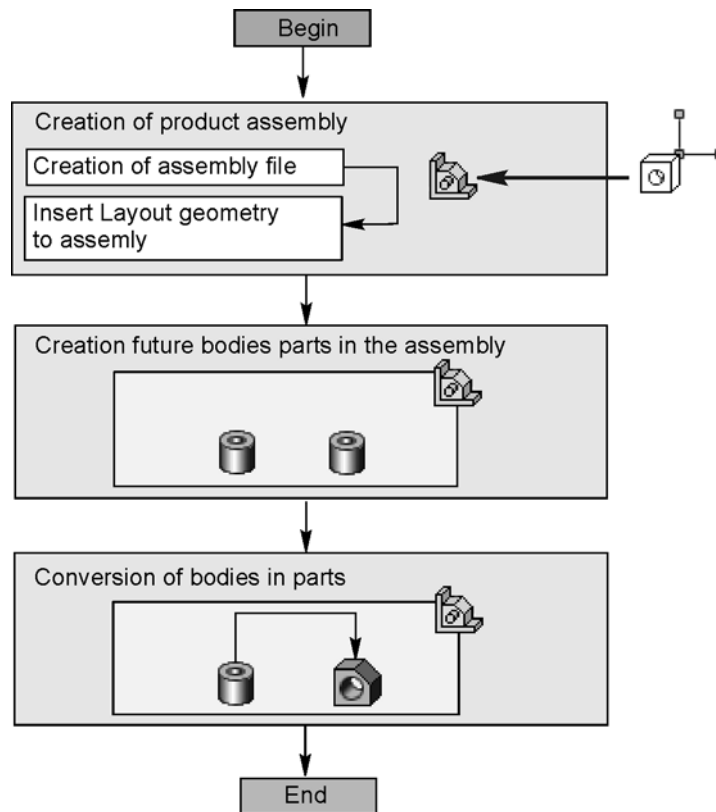
Workflow of top-down design with pre-packaging

Step	Description	Comment
1. Prepare the composition geometry	– create a file containing a part or an assembly and use this file to build the composition geometry of the assembly you are developing	– create a set of the geometric objects for the main nodes in the assembly (if necessary)

Workflow of top-down design with pre-packaging

Step	Description	Comment
2. Create an assembly and insert the composition geometry	<ul style="list-style-type: none">– create a file of the assembly you are developing;– insert the composition geometry you've prepared previously into the assembly	<ul style="list-style-type: none">– the composition geometry will then be used for managing the development process for the assembly
3. Create components within the context	<ul style="list-style-type: none">– create the components within the assembly's context	<ul style="list-style-type: none">– once all required components are created, you can close the assembly model
4. Develop components	<ul style="list-style-type: none">– open each component in a separate window;– insert the composition geometry or any of its objects into the components;– develop the component's geometry (if the component is a subassembly, then repeat the step 3 within its context)	<ul style="list-style-type: none">– use the objects of the composition geometry in the assemblies to manage the development process;– to create copies, use the sets you've prepared on step 1

The down-top method with conversion of bodies into components



Description of the down-top method with conversion of bodies into components

The method is based on creating the models of the assemblies using the bodies with subsequent conversion of these bodies into parts which will then become the components of the assembly. To create the bodies, you can use the composition geometry you've previously created and inserted into the assembly.

This method is used for designing small assemblies which primarily consist of the parts created for the first time.

Workflow of the top-down method with conversion of bodies into components

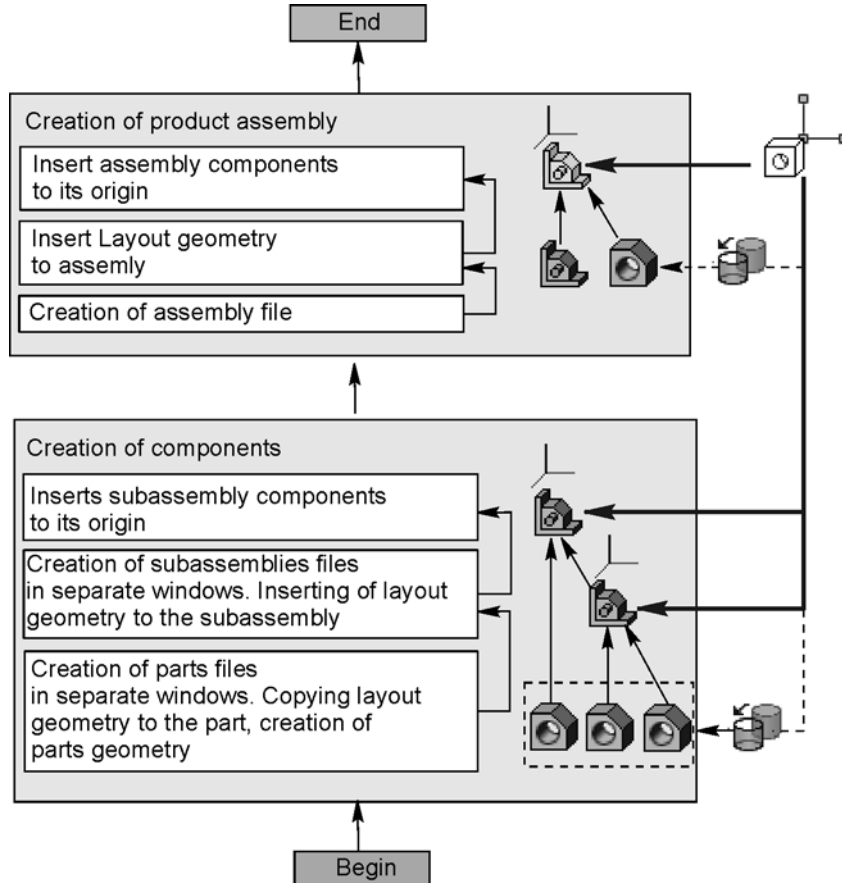
Step	Description	Comment
1. Prepare the composition geometry	– create a file containing a part or an assembly and use this file to build the composition geometry of the assembly you are developing	

Workflow of the top-down method with conversion of bodies into components

Step	Description	Comment
2. Create an assembly and insert the composition geometry	<ul style="list-style-type: none">– create a file of the assembly you are developing;– insert the composition geometry you've prepared previously into the assembly	
3. Build the bodies in the assembly	<ul style="list-style-type: none">– create the bodies of the future parts based on the objects of composition geometry within the assembly	<ul style="list-style-type: none">– create the objects in the composition geometry (planes, surfaces, etc.);– when performing the building operations, do not merge the bodies which will be used in multiple parts
4. Conversion of bodies into parts	<ul style="list-style-type: none">– the bodies in the Model Tree are incrementally transformed into parts using the Create part command¹	

¹ It is recommended to enable the **Delete original** option, otherwise the remaining bodies will be used for calculating MP parameters of the assembly's model.

The "down-top with pre-packaging" method.



Description of the down-top design with pre-packaging

In this method, the assembly's components and its subassemblies are created in separate windows, from the lower level up to the highest one. The previously-prepared objects of the composition geometry are copied into the components. The assembly, subassemblies and parts share the same zero point.

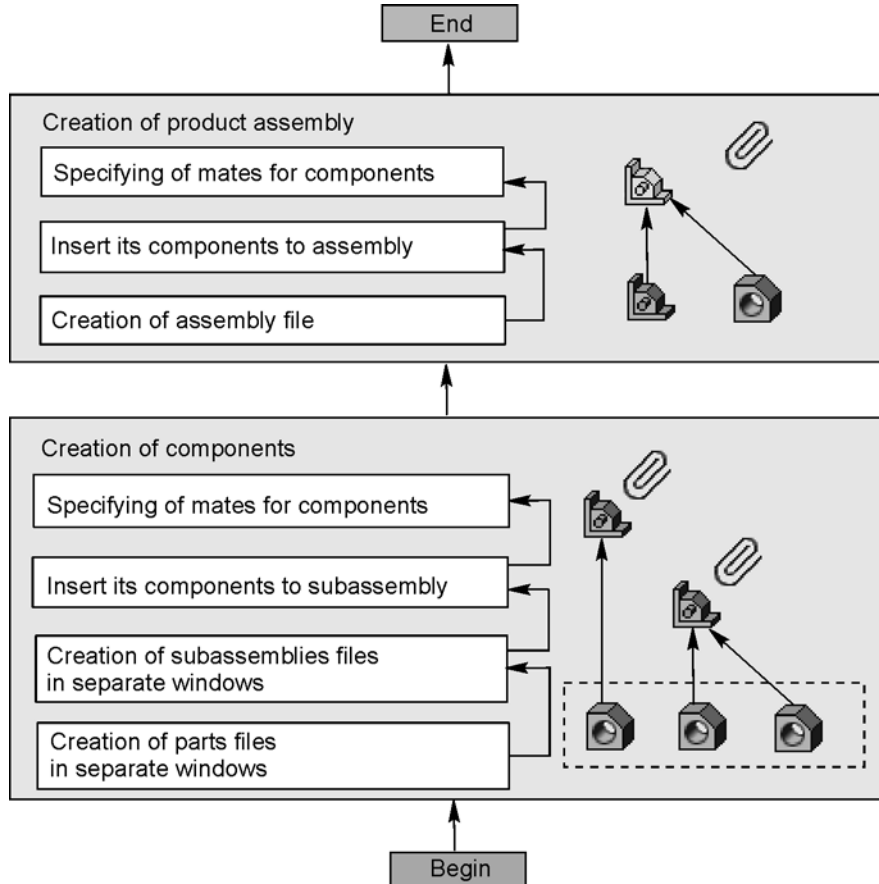
This method is used in the following cases:

- ▼ the assembly itself and its design are not known at the initial phase;
- ▼ most of parts and nodes are developed for the first time (only few 3rd party and library models are used);
- ▼ no models of the movable parts are required.

Workflow of the down-top design with pre-packaging

Step	Description	Comment
1. Prepare the composition geometry	– create a file containing a part or an assembly and use this file to build the composition geometry of the assembly you are developing	– create a set of the geometric objects for the main nodes in the assembly (if necessary)
2. Create the parts	– create files for parts; – copy the objects and (or) sets of objects required for building from the composition geometry file to the files of parts using the Copy objects command. – perform the building operations in the files of parts	
3. Create the subassemblies and insert the components into subassemblies	– create files for subassemblies; ▼ insert the composition geometry into each subassembly; ▼ insert the components into the zero point of a subassembly	– if the subassemblies are to be nested in the subassemblies of a higher level, then repeat this step for the these subassemblies of a higher level
4. Create an assembly	– create a file of the assembly you are developing; – insert the composition geometry you've prepared previously into the assembly; – insert the components into the zero point of an assembly	– by inserting the composition geometry you can better manage the assembly you are developing.

The “down-top with arrangement of components” method.



Description of the down-top design with arrangement of components

In this method, the assembly's components and its subassemblies are created step by step, from the lower level up to the highest one. The components of each assembly at each level are created and edited in separate windows. The models are created independently based on the initially established requirements to the assembly itself and its design. The components inserted in each assembly and its subassemblies can be positioned using mates.

This method is used in the following cases:









- ▼ the assembly itself and its design are not known at the initial phase;
- ▼ most of parts and nodes are ready-to-use ones (e.g. library items or objects taken from another assemblies).

Workflow of the down-top design with arrangement of components

Step	Description	Comment
1. Create the parts	<ul style="list-style-type: none">– create files for parts;– perform the building operations you need in the files of parts	
2. Create the subassemblies and insert the components into subassemblies	<ul style="list-style-type: none">– create files for subassemblies;– insert the components into the subassemblies;– apply the necessary mates to the components	<ul style="list-style-type: none">– if the subassemblies are to be nested in the subassemblies of a higher level, then repeat this step for the these subassemblies of a higher level
3. Create an assembly	<ul style="list-style-type: none">– create a file of the assembly you are developing;– insert the prepared components into the assembly;– apply the necessary mates to the components	

Legend used in workflows

Legend for workflows

	Body
	Part – a component of an assembly
	Part
	Subassembly
	Assembly
	Composition geometry
	Copy of geometry
	Local Coordinate System (LCS)



Mate

Group work on an assembly

This Chapter contains recommendations related to organization of group work with an assembly's model. These recommendations are based on the following **requirements to group work on developing the assembly**:

- ▼ developer's activities should not be duplicated;
- ▼ a developer's work with an assembly or its parts should not hinder other developers' work with the assembly and its parts;
- ▼ if necessary, a developer's work result should be protected from unauthorized changes that can be made by other developers.

These recommendations are made based on the following **development roles**:

- ▼ head of development;
- ▼ component developers.

Head of development performs the following:

- ▼ makes decisions regarding the assembly's design;
- ▼ creates the assembly file and the node model files;
- ▼ creates the baseline data, i.e. the composition geometry of an assembly;
- ▼ inserts the composition geometry into the assembly;
- ▼ places the models of its nodes in the assembly;
- ▼ creates and sets up the types of the assembly loading.

Component developers perform the following:

- ▼ create nodes and other items in an assembly;
- ▼ place the components within an assembly when required.

The best way of organizing group work is to allow the component developers perform as little operations within the assembly's context as possible.

For best results, we recommend you to do the following:

- ▼ set the password for the system loading modes (usually only the head of development knows the password);
- ▼ create the custom loading modes with unique passwords, as shown in the table.

Load Type	Purpose	Description	Comment
Development of ABC¹	To develop one or more nodes ABC in the assembly's context	<ul style="list-style-type: none"> ▼ the ABC components are assigned the Full loading mode without any limitations in editing; ▼ all other components are assigned one of the system loading modes with limited editing rights²; ▼ the component of the composition geometry is assigned the Full loading mode with the “read-only” access 	<ul style="list-style-type: none"> ▼ the loading modes are created for each developer; ▼ when working with this loading mode, a developer can edit the nodes he or she works on in the assembly's context, as well as add new components into the assembly, but cannot edit any other components in the assembly
Detailed view	for viewing an assembly with fully loaded components	<ul style="list-style-type: none"> ▼ all components along with the component of the composition geometry are assigned the Full loading mode with the “read-only” access 	<ul style="list-style-type: none"> ▼ each participant of the development is eligible to use this type
Simplified view	for viewing an assembly with simplified loading of its components	<ul style="list-style-type: none"> ▼ all components along with the component of the composition geometry are assigned the Simplified loading mode with the “read-only” access³ 	

1 Let ABC stand for one or more nodes being created by a certain developer.

2 The **Full** loading mode is assigned, when it is allowed to use the component's geometry, the **Simplified** loading mode is assigned when it is allowed to only view the component, and the **Empty** or **Size** loading mode is assigned when the component should be hidden.

3 When using the **Simplified** loading mode, the sketches, curves and points are not displayed. If you need the composition geometry objects of these types to be visible, set the **Full** loading mode to this geometry.

You can also create any custom loading modes for specific tasks (e.g. a loading mode in which all fasteners are assigned the **Empty** loading mode, or a loading mode for viewing individual nodes in an assembly where the components cannot be edited and can be viewed only in the simplified mode).

Appendix VIII. Protection of the KOMPAS-3D system

1. General information about Protection of the KOMPAS-3D system

The system KOMPAS-3D and its application, starting from version V11 and higher, are protected from unauthorized use by means of Sentitel HASP technique. The protection system is a software and hardware complex using 128-bit encryption algorithm according to Advanced Encryption Standard (AES).

1.1. Hardware protection

Standard system scope of supply KOMPAS-3D includes protection device against unauthorized use — a security dongle (Fig. 1) connected to PC USB-port. This dongle has built-in memory which stores information about paid system components KOMPAS-3D and conditions of their use.



Fig. 1. Security dongles

Hardware protection of the KOMPAS-3D system and its individual components against unauthorized use is ensured by using HASP HL keys with 3.25 version firmware.

Dongles of various types can be used to provide licensing terms flexibility (Table 1). All dongle models provide protection from unauthorized use of applications. Dongles of various types are distinguished by features of written licenses management and available memory.

Tab. 1. Types of security dongles used in the KOMPAS-3D system.

Key Type	Description	Types of licenses supported
HASP HL Pro HASP HL Max	Protection of several applications and their components on a local PC.	▼ Permanent, ▼ Component-oriented, ▼ By number of the product launches, ▼ Demo, ▼ Licensing scheme adjustable by launch counter.

Tab. 1. Types of security dongles used in the KOMPAS-3D system.

Key Type	Description	Types of licenses supported
HASP HL Time	Protection of several applications and their components on a local PC. Contains integral real-time clock.	<ul style="list-style-type: none"> ▼ Permanent, ▼ Component-oriented, ▼ Subscription (updates and new versions payment), ▼ Rental (payment for a specified period of use), ▼ Licensing scheme adjustable according to integral clock state.
HASP HL Net	Network protection for several applications and their components.	<ul style="list-style-type: none"> ▼ Permanent, ▼ Component-oriented, ▼ Floating (transferrable between users), ▼ Demo, ▼ The licensing scheme can be adjusted for the number of users and launch counter.
HASP HL NetTime	Network protection for several applications and their components. Contains integral real-time clock.	<ul style="list-style-type: none"> ▼ Permanent, ▼ Component-oriented, ▼ Floating (transferable between users), ▼ Subscription or rental, ▼ Demo, ▼ Licensing scheme adjustable by number of users, launch counter and integral clock state.

Security dongles made with HASP technology, except for HASP HL Max type dongles, are backward-compatible with HASP4 and HASP HL dongles which were used for previous KOMPAS-3D system versions protection.

HASP HL dongles can be reprogrammed so as to use all opportunities from the new technology (see section 6 на с. 2147).

1.2. Security system software implementation

Security dongle's programs are installed onto a PC during system installation KOMPAS-3D.

- ▼ Selecting menu option **Security dongle driver and administration tools** means **HASP Runtime Environment** program installation. It provides launch of software protected by HASP system and its interaction with a dongle during operation. The following HASP software components are installed automatically together with this program:
 - ▼ security dongle driver,

- ▼ **HASP Admin Control Center** — a program for network licenses management (see Section 3 на с. 2124).
- ▼ Selecting menu option **Licenses updating program** means **HASP SRM Remote Update System** program installation. It provides licenses update in installed keys in case of license agreement amendment (see section 5 на с. 2143).

2. Protection scheme

HASP allows using protected software installed on local computers or on computers included in a local network.



A firewall (e.g. Windows Firewall) may prevent HASP protection from correct working. If KOMPAS-3D starts in demo mode while key and license are available, network firewall settings will be changed.

2.1. Local security dongles

Local dongles of the following types can be used for protected application operation on a local computer:

- ▼ HASP HL Pro,
- ▼ HASP HL Max,
- ▼ HASP HL Time.

One or more of these keys are included into each workplace scope of supply according to selected licensing terms and conditions.

Information about paid components and licensing conditions is entered into local dongle's memory.

2.2. Network security dongles

HASP HL Net or HASP HL NetTime network security KOMPAS-3D dongle type is sufficient for use the system installed on computers included into local network. Information about paid components, number of licenses and licensing conditions is entered into local dongle's memory. This dongle is connected to any computer within the local network at which **HASP Run-time Environment** security program is installed.

A network dongle is included into scope of supply of several workplaces designed for networking. It allows simultaneous working of several users with the same system component. Maximum number of users working concurrently with the same component is defined by the number of licenses for this component. A computer with installed network dongle is called **network dongle server**.

Both network and local dongles can be used together for system KOMPAS-3D launch on computers connected into a local network.

2.3. Procedure for Protected Software Use

A search for active and available licenses is performed when loading the KOMPAS-3D system and/or its individual components. A local dongle is checked first. If required licenses are not found on a local dongle, they are looked for on available network dongles automatically.

If licenses are not found, the KOMPAS-3D system will be started in demo mode. System and all its components is fully functional in this mode during 30 days from the first launch.



The following will be done to start the KOMPAS-3D system and its components intentionally in demo mode:

- ▼ disconnect a local dongle,
 - ▼ turn off network dongles use function in **HASP Admin Control Center** settings (see Section 4.3 на с. 2141).
-

An informational message indicating number of days remaining to the expiration of this period will appear at each system KOMPAS-3D startup during the demo period.

The trial mode is granted one time for a particular computer.

If a dongle is not found or contains no license for the system KOMPAS-3D and/or components launched, or the number of licenses on the network dongle is exceeded, or time limit on a dongle with integral clock (HASP HL NetTime or HASP HL Time) is exceeded, the system KOMPAS-3D will work in demo mode. Depending on a certain component software implementation (3D modeling systems, some libraries etc.) it will work in demo mod or will not start.

KOMPAS-3DThe system checks periodically during work availability of a local or a network security dongle and defines if use of currently loaded system components is allowed. Dongle check is performed in background mode and causes virtually no delay in user's work. If a dongle is not found during this check, or referencing to it failed, or time limit is exceeded on a dongle with integral clock, a warning message will be displayed informing that the system shall switch to demo mode in 5 minutes. The time interval starts being counted after clicking the **OK** button in the message box. After this, the warning symbol will appear in the lower right hand corner of the application window. Five minutes later, a message will appear on the screen saying that the system is operating in demo mode, and the warning symbol will change accordingly.

User shall save all active documents and restart the KOMPAS-3D system after receiving the first message.

When working with the KOMPAS-3D system in demo mode, a number of restrictions will be applied; in particular, it will be impossible to save documents.

3. License Management if the KOMPAS-3D system is used within a network

HASP License Manager is installed on a computer together with HASP Run-time Environment. It allows licenses management during network use of the KOMPAS-3D system and its

components. Admin Control Center (hereinafter – ACC) program included into HASP Run-time Environment is used for License manager access and licenses management.

ACC default configuration provides access to all program commands and settings change. ACC launched on any network computer provides control over License manager on any computer. It is recommended to control user access to ACC resources installed on their computers.

3.1. Admin Control Center startup

3.1.1. Startup methods

Admin Control Center can be started as follows.

- ▼ By calling the main Windows menu command **Start – Programs – ASCON – KOMPAS-3D V... – Dongle software – Information about security dongles**. If this starting method is used, a tab containing information about security dongles available for this computer will open in the browser (see Section 3.3 на с. 2127).
- ▼ Select main menu command Windows **Start – Programs – ASCON – KOMPAS-3D – Security dongle programs – Info about security system**. If this starting method is used, a tab containing information about the current license manager will open in the browser (see Section 3.8 на с. 2133).
- ▼ Generally, to start ACC one should enter in the browser window (Internet Explorer, Opera etc.) a domain name or IP-address of a computer with license manager installed and port number 1947, e.g. <http://10.3.1.37:1947> or http://LM_server:1947 and go to this address.



Port 1947 shall be open, otherwise ACC cannot be used.



The following conditions shall be met to get access to license manager on a remote computer:

- ▼ remote user access shall be allowed in ACC settings on the remote computer (see section 4.4 на с. 2142),
 - ▼ a password for ACC access on this computer shall be known to change ACC settings on the remote computer (see section *ACC password protection* на с. 2139).
-

The following shall be entered in the address line to get access to license manager on a local computer: <http://localhost:1947>.



When using Window 8 with the metro version of Internet Explorer and default settings, access to the license manager on the local machine will not be available. It is due to operation of the Enhanced Protected Mode (EPM). By default, this mechanism blocks addressing to <http://localhost>.

If you need to use the <http://localhost:1947> address with the metro version of IE, you can add it to the list of reliable sites or modify the browser's setting, i.e. disable identification of addresses as intranet ones.

3.2. ACC interface

Default browser window with open program page will appear on the screen after ACC launch. In Fig. 2, you can see the Internet Explorer window after calling the **Start — Programs — ASCON — KOMPAS-3D V... — Dongle software — Information about security dongles** command.

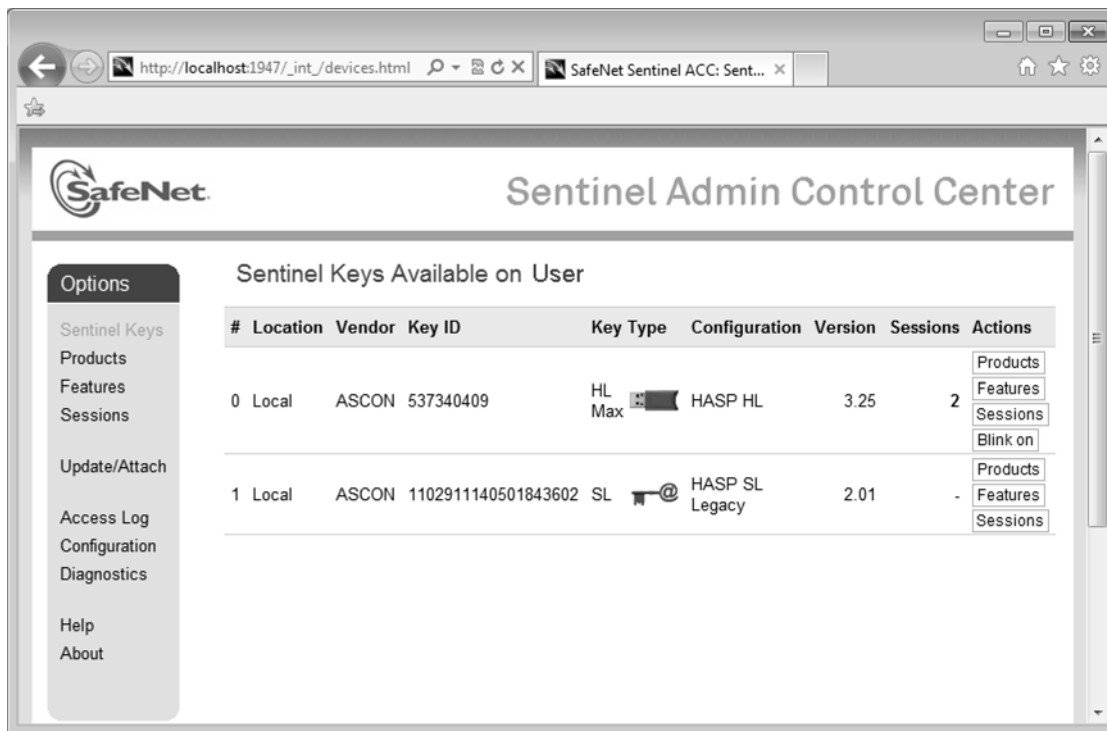


Fig. 2. ACC page in the browser

ACC commands menu is displayed in the left part of the page. Description of commands is presented in Table 2. These commands are related to computer License manager those network name or IP-address is shown in ACCC header line (hereinafter referred as *current computer*). A new tab appears in the browser window after calling this command. Control elements on this tab provide additional actions related to this command.

Tab. 2. Admin Control Center command description

Command name	Command purpose
Sentinel Keys (Sentinel)	Displays a list of security dongles available in the network, including both network and local dongles.

Tab. 2. Admin Control Center command description

Command name	Command purpose
Products (Products)	Displays a list of all applications available in the network using all License managers.
Components (Features)	Displays the following information: <ul style="list-style-type: none"> ▼ a list of the KOMPAS-3D system components licensed for each dongle, including both network and local dongles, ▼ components licensing terms and conditions, ▼ number of users for each component.
Sessions (Sessions)	Displays client sessions on the current computer, both local and connected to License manager on this computer via network. These sessions can be terminated forcibly, if necessary.
(Update/Attach) (Update/Attach)	Ensures license update on a dongle.
Access log (Access Log)	Displays a log of connections to License manager on the current computer. This log can be stored in the text file <i>access.log</i> created automatically in the same folder as ACC settings file <i>hasplm.ini</i> . A complete path to this file is displayed in the lower part of the browser tab at ACC setup page (see section4 на с. 2135).
Configuration (Configuration)	Allows setting up ACC use parameters on the current computer, e.g., user access to ACC control, access to remote License manager from the current computer, access to the current computer License manager for remote users, format of created log files (see Section4 на с. 2135).
Diagnostics (Diagnostics)	Allows viewing information about the current License manager and prepare a report for technical support service.
Help (Help)	Provides access to ACC help system.
About this Software (About)	Provides information about License manager version and a link to the site of HASP system developer's knowledge base.

A link to ACC help section related to the corresponding tab is located in the lower right corner of each tab.

3.3. Viewing a list of dongles available in the network

In order to view the list of network and local hardware keys that are connected to the network's machines, call the following command: **Sentinel Keys (Sentinel Keys)**.

The **Sentinel Keys tab will appear in the browser window (Sentinel Keys available on) <current machine's name>**. This tab displays a table containing information about dongles. Description of this table is given in Table 3.

Tab. 3. List of dongles available in the network

Column Name	Column contents
PC (Location)	Name of a computer with connected dongle. If a dongle is connected to the current computer, its name will be displayed as <i>Local</i> . Remote computer name is a link. A referenced computer becomes the current one after going to this link. ACC of this computer will be open on a new tab. Remote user access shall be allowed in ACC settings on the remote computer.
Vendor Vendor	Software vendor code.
Key ID (Key ID)	Dongle unique identifier.
Key Type (Key Type)	Security dongle type designation and its reduced image.
Configuration (Configuration)	Information about the key's configuration.
Version (Version)	Dongle firmware version number.
Sessions (Sessions)	Number of open access sessions for the dongle.

Tab. 3. List of dongles available in the network

Column Name	Column contents
Actions (Actions)	<p>Commands providing access to additional information about a dongle. Command set depends on if the dongle is network or local one.</p> <ul style="list-style-type: none"> ▼ Sessions — allows opening a tab with information about sessions for this dongle. ▼ Components (Features) — allows opening a tab with information about application components those licenses are available on this dongle. Available for the current computer local dongle. ▼ LED on/off Blink on/off — allows dongle LED blinking control to ensure its identification. ▼ Browse — allows viewing all application components for a selected network dongle. License manager installed o a computer to which this dongle is connected will be open in a new browser tab. Access to remote license manager is possible if remote user' access is allowed in its settings (see section 4.1 на с. 2135). ▼ Net Features Network Features — allows viewing application components for the selected network dongle available for the current computer.



A local security software key is displayed first in the list of available network and local security dongles. This key ensures operation of the system KOMPAS-3D and its components during the trial period. The **Licenses** command allows viewing information about this period:

- ▼ state (is trial run is started or not, is trial run is possible or its period is expired),
- ▼ start and termination date and time.

3.4. Viewing a complete list of applications available for the current computer in the network

To view application lists call **Products** command.

Products available on <current computer name> tab will appear in the browser window.

This tab displays a table with designations of applications related to all License managers in the network. Description of this table is given in Table 4.

Tab. 4. List of applications available for the current computer in the network

Column Name	Column contents
Product name (Product Name)	Application name defined by its vendor.

Tab. 4. List of applications available for the current computer in the network

Column Name	Column contents
Vendor Vendor	Software vendor code.
PC (Location)	Name of a computer with connected dongle for this component. If a dongle is connected to the current computer, its name will be displayed as <i>Local</i> .
Actions (Actions)	Commands providing access to additional information about the application. ▼ Components (Features) — allows opening Licenses for tab displaying application components list.

3.5. Viewing application components list

The **Components (Features)** command can be called to view the list of application components licensed on dongles available in the network.

Licenses for Features available on <current computer name> tab will appear in the browser window. This tab displays a table containing information about application components licensed on each dongle (network and local) available in the network. The table contains information about licensing conditions and current components use. Description of this table is given in Table 5.

Tab. 5. The list of application components licensed in the keys available in the network.

Column Name	Column contents
#	Serial number of the line
Vendor Vendor	Software vendor code.
Key ID (Key)	Dongle unique identifier.
Product (Product)	Unique identifier and product's name specified by the vendor.
Component (Feature)	Application component unique identification number and name assigned by its vendor.
PC (Location)	Name of a computer with connected dongle. If a dongle is connected to the current computer, its name will be displayed as <i>Local</i> .

Tab. 5. The list of application components licensed in the keys available in the network.

Column Name	Column contents
Access (Access)	Type of computers for which access for use this component is allowed. The following options are possible: <ul style="list-style-type: none"> ▼ <i>Loc</i> — access is allowed for local computer only, ▼ <i>Net</i> — access is allowed for remote network computers, ▼ <i>Disp</i> — access for remote computers is allowed via terminal server (not used in the KOMPAS-3D system).
Counter (Counting)	Method of component use quantity count. The following options are possible: <ul style="list-style-type: none"> ▼ <i>Process</i> — all access requests for the same process use are considered as one access, ▼ <i>Station</i> — all access requests for the same computer use are considered as one access, ▼ <i>Login</i> — all requests for use are included in the count of component uses.
Sessions (Logins)	Number of users currently using this application component.
Limit (Limit)	Maximum number of users allowed for concurrent component use.
Detached (Detached)	Currently not used.
Restrictions (Restrictions)	Restrictions related to the application component use on this dongle. E.g., <i>Expired</i> — license validity period has expired on a dongle with integral real-time clock.
Sessions (Sessions)	Number of current dongle access sessions.
Actions (Actions)	Commands providing access to additional information about the application. <ul style="list-style-type: none"> ▼ Sessions — allows opening Sessions on tab containing information about access sessions for a certain application components.

3.6. Viewing the list of access sessions to protected products and session management

In order to view the list of access sessions, call the **Sessions** command.

Sessions on <current computer name> tab will appear in the browser window. This page displays a table containing information about all access sessions from local and remote users to the current computer. Control elements located on the page support viewing information about access sessions and their termination.

Description of this table is given in Table 6.

Tab. 6. List of access sessions to the current computer.

Column Name	Column contents
ID	Unique session identifier.
Key (Key)	Dongle unique identifier.
PC (Location)	Name or IP-address of a computer with connected dongle. If a dongle is connected to the current computer, its name will be displayed as <i>Local</i> .
Product (Product)	Unique identifier and product's name specified by the vendor.
Component (Feature)	Application component unique identification number and name assigned by its vendor.
Address (Address)	IP-address of a computer from which access is performed, or <i>Local</i> , if accessed from the local computer.
User (User)	A name of a user who uses this application component.
Client (Machine)	A network name of a computer from which the application component is used and ID of a process which opened the access session.
Login Time (Login Time)	Time of beginning access session to the application component.
Timeout (Timeout)	Currently remaining period of license holding at the server. Initial time interval value is 12 hours. The value in this column becomes equal to initial one at the moment of license availability check, i.e. each 15 minutes. If the KOMPAS-3D system is shut down abnormally, license check will not be performed. If the KOMPAS-3D system operation at this workplace would not be resumed, the license will be turned off after column value zeroing.

Tab. 6. List of access sessions to the current computer.

Column Name	Column contents
Actions (Actions)	<p>Commands providing access to additional information about the application.</p> <ul style="list-style-type: none"> ▼ Disconnect — terminates access session to the current application component for the current user (disconnects a user from the license). To run this command one shall know ACC access password for a computer to which a security dongle is connected (see section <i>ACC password protection</i> на с. 2139).

3.7. Viewing a log of connections to License manager on the current computer

Access Log command shall be called to view connections log. **Access Log for <current computer name>** page will appear in the browser window. This page displays a table containing information about all access sessions from local and remote users to the current computer License manager. The latest 20 entries are displayed in the table by default. Buttons **20**, **100** and **1000** are designed to select the number of entries displayed per page.

Each log entry includes the following default information:

- ▼ entry creation date and time,
- ▼ user IP-address and port,
- ▼ user ID,
- ▼ access method,
- ▼ URL of the resource to which the request is addressed,
- ▼ function used,
- ▼ function parameters,
- ▼ value returned by the function.

The default log template can be changed at **Basic Settings** tab at ACC setup page (see section *Log file template* на с. 2137).

If option **Write an Access Log File** is turned on at **General settings** tab at ACC configuring page, the connection log is saved in the text file *access.log*. This file is created automatically in the same folder as ACC settings file *hasplm.ini*. A complete path to this file is displayed in the lower part of the browser tab at ACC setup page (see section4 на с. 2135). Files are saved in the folder *C:\Program Files\Common Files\Aladdin Shared\HASP* by default.

3.8. Viewing the information about the current License Manager.

To view information about the current License manager call **Diagnostics** command.

The **Diagnostics for Sentinel License Manager on <current computer name>** tab will appear in the browser window. This tab displays a table containing information about the License manager.

Description of this table is given in Table 7.

Tab. 7. License manager information

Description of information	Contents
License manager version (License Manager Version)	Current License manager version
Computer name (Computer Name)	The name of the computer at which License manager is installed, and process identifier (PID).
Operating system (Host Operating System)	The name and version of an operating system of the computer at which License manager is running.
Protocols (LM Protocols)	<ul style="list-style-type: none">▼ The current protocol used by License manager. Possible values are IPv4 (IPv4 only) or IPv4, IPv6 (IPv4 and IPv6).▼ Current License manager IP-address.
Uptime (Uptime)	Duration of the current access session to the License manager.
Template Sets (Template Sets)	A list of ACC interface accessible templates.
Current Template (Current Template)	Name, version number and development date of the interface template that is currently in use.
Current usage (Current Usage)	Information about License manager current usage: <ul style="list-style-type: none">▼ logins — the number of licenses captured,▼ sessions — the number of current access sessions to License manager,▼ connections — the number of current network connections from the total available.
Sessions (Login Requests)	The number of licenses obtained from the current License manager since its startup.
Requests (Requests)	The number of requests to the current License manager since its startup.
Data Volume (Data Volume)	The number of information bytes received and transmitted by this server since the License manager startup.

Tab. 7. License manager information

Description of information	Contents
Errors (Errors)	Total number of errors related to the dongle or transmission on this server since the License manager startup.
Client Threads (Client Threads)	The number of simultaneous sub-processes opened by the License manager.
Run-time Environment (Run-time)	A list of launched HASP system components and their versions.

Beneath the table with information about the License Manager, there is a button for calling the **Generate Report** command. This command generates diagnosis report in HTML format. This report contains detailed information about the system at which this copy of License manager is running, about licenses use, and other information that can be used for diagnosis purposes. This report should be used when calling the technical support service.

4. ACC setup

ACC setup allows setting the following parameters:

- ▼ user access rights for security system network resources,
- ▼ parameters of remote License managers access,
- ▼ network computers' users access rights to control License manager on the current computer.

The **Configuration** command is called to perform setup. The **Configuration for Sentinel License Manager on <current computer name>** tab will appear in the browser window. Control elements designed for configuring are grouped on tabs. Tab names correspond to settings type.



An administrator's password shall be entered to proceed with ACC configuration, if it has not been set previously (see section *ACC password protection* на с. 2139).

ACC settings are saved in *hasplm.ini* file which is created automatically at first change of the default settings. A complete path to the file is displayed in the lower part of the browser tab.

4.1. ACC basic settings; Basic Settings tab

4.1.1. ACC basic settings

ACC basic settings include the name of a computer at which ACC is being set up, log creation parameters and password protection settings. The tab controls are described in Table 8.

Tab. 8. Control elements on the **Basic settings** tab

Control element name	Description
Computer name (Machine Name)	Network name of the computer at which ACC is being set up
Allow remote access to ACC (Allow Remote Access to ACC)	This option allows control over remote users access to ACC of the computer those name is entered in Computer name field. This option is disabled by default.
Display Refresh Time (Display Refresh Time)	Information refresh period at ACC tabs in seconds
Table Rows per Page (Table Rows per Page)	The number of table rows displayed at each page of the tab. The value may be between 5 and 100.
Write an Access Log File (Write an Access Log File)	This option supports creation of License manager access log file. If this option is activated, the following options controlling log file creation become available: Include Local Requests, Include Remote Requests, Include Administration Requests.
Include Local Requests	Allows adding to log information about local computer users access sessions to application components licensed on a dongle connected to this computer.
Include Remote Requests	Allows adding to log information about network computer users access sessions to application components licensed on a dongle connected to this computer.
Include Administration Requests	Allows adding to the log information about access sessions to License manager using ACC.
Write an Error Log File (Write an Error Log File)	This option supports error log file control.

Tab. 8. Control elements on the **Basic settings** tab

Control element name	Description
Write Log Files Daily (Write Log Files Daily)	This option makes it possible to create an individual log file for every day during which any log entries took place.
Days Before Compressing Log Files (Days Before Compressing Log Files)	You can use this option to specify the number of days upon expiration of which the log files will be compressed. Compression is performed using the standard ZIP algorithm. The <i>zip</i> extension is added to the file's name. The maximum number of days is 9999. If the value is equal to zero, files will not be compressed.
Days Before Deleting Log Files (Days Before Deleting Log Files)	You can specify the number of days during which the closed log file (compressed or uncompressed) will be saved on the computer disk. Upon expiration of the period, the file will be deleted automatically. If the value is zero, log files will not be deleted automatically.
Write a Process ID (.pid) File (Write a Process ID (.pid) File)	This option supports process ID file.
Password Protection (Password Protection)	You can set a password for access to ACC. Specify, if only Configuration Pages or All ACC Pages should be protected. In order to set or change password, click the Change Password button.

4.1.2. Log file template

The **Edit Log Parameters** command allows changing template of License manager access log template.

The **Edit Log Parameters** page appears in the browser window after **Edit log parameters** command calling. A field in the upper part of the page contains the current set of template elements designations. Field content is presented in a text form. Elements' designations are reserved words. They are placed between curly braces. Comments can be added to log elements for clarification. Elements designations can be edited as plain text or using content of the field **Tags available for log: (Available tags for log:)**.

The **Available tags for log** field: contains designations and brief descriptions for available template elements. To add an element top the template it shall be selected and then **Add** command shall be called. Element's designation will be added to the end of the list.

The **Back to Configuration** command completes template editing and returns a user to configuration page.

An example of generated log template is shown in Fig.3.



Fig. 3. An example of generated template

According to this template, each log entry shall contain the following information:

- ▼ entry creation date and time,
- ▼ identifier of a dongle whose license is used,
- ▼ user ID,
- ▼ license use session duration,
- ▼ component ID,
- ▼ request type, e.g., license obtaining (LOGIN), license release (LOGOUT).
- ▼ license server IP-address,
- ▼ request running result code.

Element **newline** performs line change in the log.



Request running result code can be used to analyze possible faults of request running. For example, code value “0” means successful request execution. Code value “7” is returned if a security dongle is not found.

Log file fragment generated using a template shown in Fig.3 is given below.

2009-01-23 11:30:00 AM HASP ID:1086818230 User ID:Test@Tester Session duration: 0 days 0 hours 0 minutes 0 seconds Component ID:120 Request type:LOGIN License server IP: 127.0.0.1 Result code:0

2009-01-23 11:31:54 AM HASP ID:1086818230 User ID:Test@Tester Session duration: 0 days 0 hours 1 minutes 54 seconds Component ID:120 Request type:LOGOUT License server IP: 10.3.1.2 Result code:0



Besides records with information about licenses usage by users, log file contains many other data. Log file content filtration, e.g. in a text editor, is reasonable for convenient content analysis.

4.1.3. ACC password protection

The **Change Password** command allows setting ACC password.

The following actions are password protected during HASP software operation:

- ▼ user disconnection from license (see Section 3.6 на с. 2131),
- ▼ ACC configuration change.

The **Change Password** command is called to set a password. The **Change Password** page appears in the browser window. The current password should be entered in the **Current Admin Password** field.



No password is set by default. The **Current Admin Password** field should be left blank during first password setting.

Enter the new password in **New Admin Password** field; then enter it again in **Re-enter new Admin Password** field. Call the **Submit** command after setting a new password. Call **Cancel** command to reject changes.

Password setting tab will be closed. **Basic Settings** tab becomes active.



Previous password will remain active during browser work session. Restart browser to activate changes.

To activate changes made on **Basic settings** tab call **Submit** command. The **Set Defaults** command returns all settings to default values.



The **Set Defaults** command does not apply to the set password.

4.2. Users tab

Settings made on **Users** tab allow direct setting names of users who are allowed or not allowed to access License managers and names of computers with installed License managers access to which is attempted.

The tab controls are described in Table 9.

Tab. 9. Control elements on the **Users** tab

Control element name	Description
Access restrictions (User Restrictions)	This field sets permissions/restrictions rules applied for users' attempts to access License manager.

Rules have the following format:

`<restriction>=[username]@[hostname]`

Parameters description is given in Table 10.

Tab. 10. Access control rules' elements

Parameter designation	Name	Possible values	Description
restriction	Restriction type	allow	allowed
		deny	denied (prohibited)
hostname	Computer name	test-2	
		all	all computers in the network
		none	none computer in the network
username	User name	User1, testuser	
		all	all users in the network
		none	none of users

Parameters *hostname* and *username* are optional. If a parameter is not entered during the line entering, its value becomes equal to *all*.

E.g., if a rule of the kind *allow=[username]* is set, access to License manager for the user *[username]* will be permitted no matter on which network computer the License manager is installed.

If a line is set in the form of *allow=[username]* during input, it will be changed after configuration change submitting using **Submit** command to the type *allow=[username]@all*.

Similarly, if a line is set in the form of *allow=@[hostname]* during input, access to License manager installed on the computer *[hostname]* will be permitted no matter which user tries to access.

If a line is set in the form of *allow=@[hostname]* during input, it will be changed after configuration change submitting using **Submit** command to the type *allow=all@[hostname]*.

Each rule shall be written in a separate line. Rules are processed one after another top-bottom. Rules processing is stopped when the first conditions conformity is found.

Examples of rules processing are given in Table 11. It is assumed that all rules are written in the **User Restriction** field in the same order as in the table.

Tab. 11. Examples of access control rules processing

Rule	Description of rule processing performed by ACC
deny=User1@seat1	User <i>User1</i> has no permission to access License manager installed on the computer <i>seat1</i> .

Tab. 11. Examples of access control rules processing

Rule	Description of rule processing performed by ACC
allow=User1@all	User <i>User1</i> has access to all computers except <i>seat1</i> . Prohibition is defined by the previous rule.
allow=User2@all	User <i>User2</i> has access to all computers.
deny=all@seat2 deny=all@seat3 deny=all@seat4	Access to License managers installed on computers <i>seat2</i> , <i>seat3</i> , <i>seat4</i> is prohibited for all users except <i>User1</i> and <i>User2</i> . Access rules for these users are already processed.

The **Show Recent Users** command displays a list of users last accessed License managers. To activate settings changes made on this tab call **Submit** command. **Cancel** command rejects settings change. The **Set Defaults** command returns all settings to default values.

4.3. Remote License managers access tab, (Access to Remote License Managers)

Control elements located on **Access to remote License Managers** tab allow setting names of computers with License managers installed to which users have access.

The tab controls are described in Table 12.

Tab. 12. **Remote License Managers Access** tab control elements

Control element name	Description
Allow Access to Remote Licenses (Allow Access to Remote Licenses)	This option ensures control accessing to License managers installed on other computers in the network from the current computer. By default, the option is enabled.
Broadcast Search for Remote Licenses (Broadcast Search for Remote Licenses)	This option controls searching computers with installed License managers in the network. If this option is disabled, names of computers among which License managers will be searched shall be specified explicitly in Specify Search Parameters field. If this option is turned on, search on all computers will be performed (broadcast search).
Aggressive Search (Aggressive Search for Remote Licenses)	This option controls the way of searching computers with installed License managers. If this option is turned on, remote License managers will be accessible even if they could not be found using standard search tools in UDP protocol. The "Aggressive" search method reduces information refreshment rate about HASP system state, but it may permit firewalls bypassing.

Tab. 12. **Remote License Managers Access** tab control elements

Control element name	Description
Search Parameters (Specify Search Parameters)	<p>This field allows explicit specifying computer names for License managers search. Computer addresses can be specified as follows:</p> <ul style="list-style-type: none"> ▼ Computer IP-address, e.g. 10.3.1.37; ▼ computer name in the network, e.g. test-2; ▼ broadcast address, e.g. 10.3.1.255. <p>If Ipv6 protocol is used, addresses shall be written in the format of this protocol. Each IP-address shall be on a separate line.</p>

To activate settings changes made on this tab call **Submit** command. **Cancel** command reverts settings change. The **Set Defaults** command returns all settings to default values.

4.4. Access from Remote Clients tab

Control elements located on **Access from remote clients** tab allow setting up the following parameters:

- ▼ names of computers whose users are allowed or not allowed to access License manager on the current computer,
- ▼ License manager access rules.

The tab controls are described in Table 13.

Tab. 13. **Access from Remote Clients** tab control elements

Control element name	Description
Access from Remote Clients (Allow access from Remote Clients)	This option controls remote users access to License manager on the current computer.
Access restrictions (Access Restrictions)	This field sets permissions/restrictions rules applied for users' attempts to access License manager.

Rules have the following format:

restriction=[item]

Parameters description is given in Table 14.

Tab. 14.

Parameter designation	Name	Possible values	Description
restriction	Restriction type.	allow	allowed
		deny	denied (prohibited)
item	Computer IP-address or network name	10.3.1.27 or TEST2	
		all	all computers in the network
		none	none computer in the network

Each rule shall be written in a separate line. Rules are processed one after another top-bottom. Rules processing is stopped when the first conditions conformity is found.

The **Show Recent Client Access** command allows viewing a list of computers from which License manager on the current computer was accessed recently.

To activate settings changes made on this tab call **Submit** command. **Cancel** command rejects settings change. The **Set Defaults** command returns all settings to default values.

5. Security dongle remote reprogramming

A dongle can be reprogrammed remotely using system remote update program HASP Remote Update System.

5.1. General procedure for licenses update

You will receive network or local security dongles when purchasing KOMPAS software. Dongles memory contains information about KOMPAS modules which are paid for and, consequently, may be operated by the user.

You may later need to change licensing conditions, e.g., buy additional system modules KOMPAS-3D and install them on the same computer, change the number of licenses etc.

The following shall be done to change license conditions:

1. Make a contract on license conditions change.
2. Generate dongle status file containing information about user licenses state.
3. Send this status file to ASKON company via e-mail.
4. Pay for components to be added.
5. Receive a reply file from ASKON company.
6. Reprogram the dongle by writing in its memory information about newly purchased modules.
7. Install paid components of the KOMPAS-3D system.

5.2. Dongle status file generation

Dongle status file generation and dongle reprogramming are performed using HASP SRM Remote Update System (hereinafter HASP SRM RUS) program. The executable file is *hasprusa.exe*.

In order to run the HASP SRM RUS, run the main Windows menu command: **Start — Programs — ASCON — KOMPAS-3D V... — Security dongle software — License update**. Also, the *hasprusa.exe* file located in the \HASP folder the main folder of KOMPAS-3D may be launched.

HASP SRM RUS window (Fig.4) appears on the display after program startup.

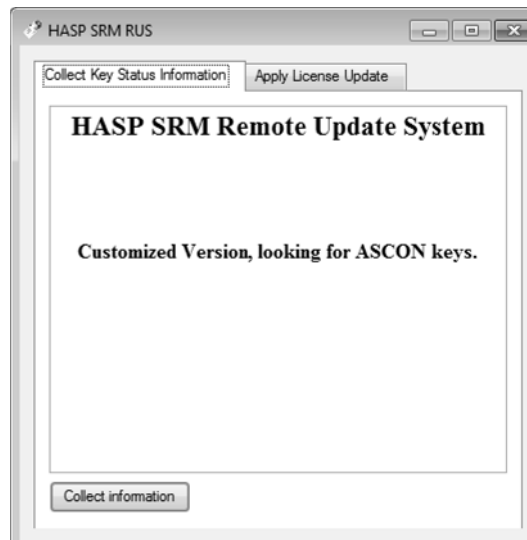


Fig. 4. License Updater window; data collection tab

Tab of data collection about dongle licenses status information **Collect Key Status Information** is open by default.

To prepare dongle status file please do the following.

1. Insert a dongle into computer port connection.
2. Click the **Collect information** button.

The standard Windows save file dialog will appear on the screen. Dongle status file has the default extension *c2v* (from *customer to vendor*).

3. Enter a request file name and close the dialog window.

A message about successful operation completion *Key status retrieved from HASP successfully* will appear in the program window. Generated file will be saved in the specified folder.

If the program cannot find a dongle during operation, a warning message will be displayed (Fig.5).

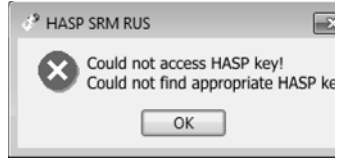


Fig. 5. HASP SRM RUS message in case no key is found

In this case insert security dongle into USB-port and repeat operations.

If the program finds several dongles during operation, **Select HASP** dialog will be displayed (Fig.6).

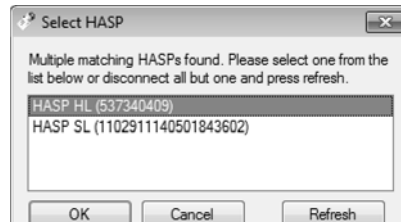


Fig. 6. HASP SRM RUS message if several keys are found

In this case select a single dongle symbol with a mouse and click **OK** or disconnect all dongles except the necessary one and click **Refresh** button.



If licenses shall be updated on several dongles, the above operations shall be performed for each dongle in turn. Status file will be generated for each dongle.

5.3. Sending status file

Generated dongles status files shall be sent to ASKON company via e-mail with necessary comments.

We recommend you to contact the same office where software from ASKON company was initially purchased. If you purchased your KOMPAS-3D system from a regional dealer, you may update dongles with his aid.

5.4. Dongle reprogramming after receiving an answer.

After payment for additional KOMPAS-3D system modules, ASKON Company will send you files with license updates.

Update files can be supplied in the following formats:

- ▼ file with the *v2c* extension (from *vendor* to *customer*).
- ▼ executable file with *exe* extension.

To reprogram a dongle using *v2c* please do the following.

1. Insert a dongle into computer port connection.
2. Run HASP SRM RUS and open **Apply License Update** tab in the displayed program window (Fig. 7).

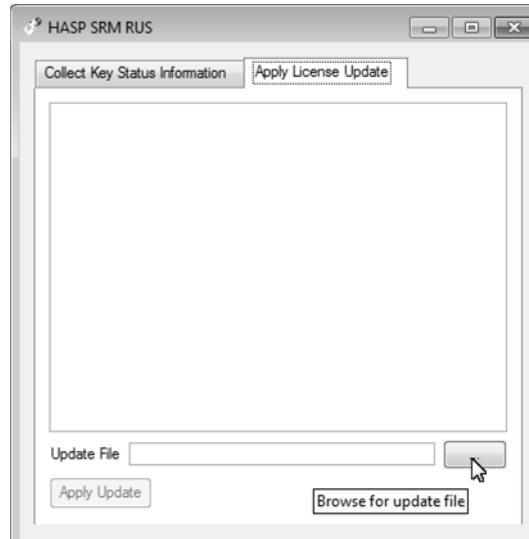


Fig. 7. Window of the License Update Program; License Update page

3. Click update file search button **Browse for update file**. The standard Windows Open File dialog will appear on the screen.
4. Open the update file received from ASKON Company.
5. Click the **Apply Update** button.

Data on additional products contained in the license update file and corresponding to the current dongle will be written into this dongle. If data are written successfully in a dongle, a confirmation message will be displayed.



If licenses shall be updated on several dongles, the above operations shall be performed for each dongle in turn.

If an update file is sent by the vendor as an executable file with *exe* extension, this file shall be launched for execution to update licenses. HASP SRM RUS will launch automatically.

5.5. KOMPAS-3D system components installation

After security dongle reprogramming, the KOMPAS-3D system components, to which licenses have been obtained, can be installed. For this, do the following.

- ▼ Launch the Basic package installation.
- ▼ In **Program service** dialog of the Installation wizard, select **Change**.
- ▼ Specify necessary components in further dialogues of the Wizard and install them.

6. Dongle firmware update

HASP Run-time Environment software version 5.95 supplied with KOMPAS-3D V15 automatically updates firmware of HASP HL security dongles obtained at previous versions purchase to version 3.25 of KOMPAS-3D. Update is performed after dongle connection to USB port.

Furthermore, “outdated” dongles firmware can be updated to version 3.21. For this purpose an update program with executable file *FirmwareUpdate.exe* shall be used. This file is located in *KOMPAS-3D_V...\KOMPAS-3D\Support* folder on the installation disk.

Connect a dongle whose firmware is to be updated.

Open an executable file to launch the program. The **HASP SRM RUS** window will pop up (Fig.8).

Apply Update button is designed for security dongle firmware update. The program window shall be closed after firmware update.



Fig. 8. Firmware update program window

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